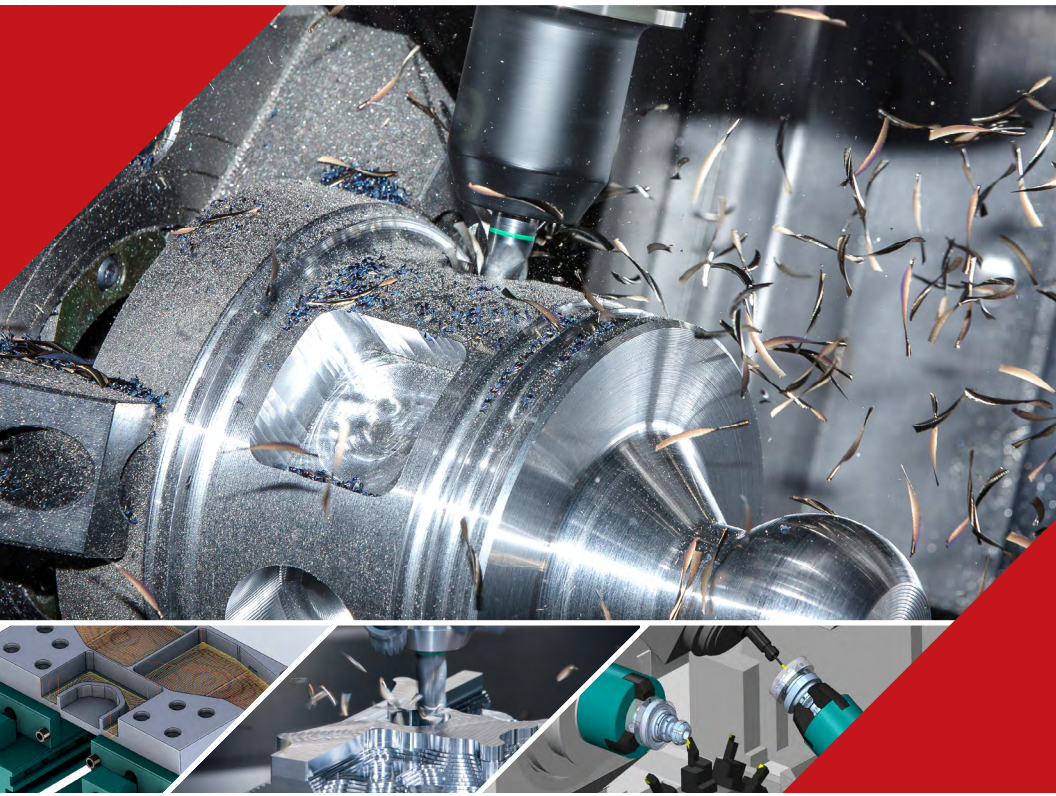
SolidCAM 2018

The unique, revolutionary Milling technology

iMachining® patent by SolidCAM

TIME SAVINGS 70% ... AND MORE!



SolidCAM 2018 Modules Overview

Parts & Recordings

iMachining 2D & 3D | 2.5D Milling | HSS | HSM | Indexial Multi-Sided | Simultaneous 5-Axis | Turning & Mill-Turn | Solid Probe

CERTIFIED Gold Product SOLIDWORKS

SolidCAM + SOLIDWORKS

The Complete Integrated Manufacturing Solution

SolidCAM

iMachining – The Revolution in CAM!

Table of Contents

SolidCAM 2018. 2
iMachining 2D 6
iMachining 3D 14
2.5D Milling 20
2.5D Milling – ToolBox Cycles 28
2.5D Milling – Feature Recognition 30
High Speed Surface Machining (HSS) 32
3D Mill High Speed Machining (HSR/HSM) 40
Indexial Multiaxis Machining 50
Simultaneous 5-Axis Machining 58
Turning 72
Mill-Turn 82
Solid Probe 94
System Requirements 100

Document number: SCMOPRENG1_2018

About this booklet

This Modules Overview booklet is designed to provide you with an overview of the available modules offered by SolidCAM. In addition to the information about each module, there are various SolidCAM parts that demonstrate the use and features of the available modules.

The parts are included with the installation of SolidCAM and are located on the hard drive of your computer (full path: **C:\Users\Public\Documents\SolidCAM\SolidCAM2018\User\Getting_Started_Examples\SW**).

Using the interactive features in this booklet

The various parts can be automatically opened using the interactive features in this booklet. To enable the correct loading of the parts, make sure they are always stored in the default installation location. For each part, there is a corresponding SolidCAM Professor recording that can also be viewed.

To use the interactive features in this booklet:

- Click the icon to load the example part.
- Click on the part image to open and play the recording in YouTube.

or

- Click to download and store the recording on your hard drive.

The recordings for this booklet were made using previous and current versions of SolidCAM integrated with SolidWorks.

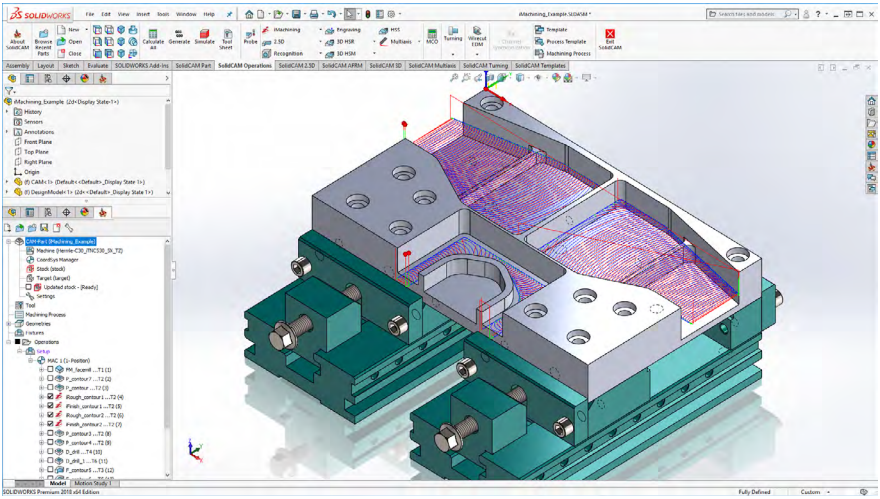
Adobe Reader

It is highly recommended to use Adobe Reader when viewing the PDF version of this booklet. If you do not already have Adobe Reader, it is a free PDF viewer that can be downloaded from Adobe's website via this link – https://get.adobe.com/reader/. By default, Adobe Reader allows the launching of external files and connecting to URLs, which is required for the interactive features to function properly. If you prefer using a different PDF viewer, it is important to make sure that your preferred Reader allows the executing of external commands.

To see many more Professor videos that cover all the features and uses of SolidCAM, visit our website at www.solidcam.com/professor.

For the most up-to-date PDF version of this Modules Overview booklet, visit our website at www.solidcam.com/subscription/documentation.

SolidCAM is the Complete Manufacturing Solution Integrated in SOLIDWORKS



SolidCAM is the Leading, ‘Best-in-Class’ CAM Suite for Profitable CNC Programming in SOLIDWORKS

SolidCAM, including the revolutionary iMachining, is seamlessly integrated in SolidWorks with full tool path associativity to the SolidWorks model. With the single-window integration in SolidWorks, all machining operations can be defined, calculated and verified without leaving the parametric SolidWorks assembly environment.

SolidCAM + SOLIDWORKS provides a powerful, easy-to-use integrated CAD/CAM solution that support the complete range of major manufacturing applications including iMachining 2D, iMachining 3D, 2.5D Milling, High Speed Surface Machining (HSS), 3D Mill High Speed Machining (HSR/HSM), Indexial Multiaxis Machining, Simultaneous 5-Axis Machining, Turning, Mill-Turn and even Solid Probe.

All 2D and 3D geometries used for machining are fully associative to the SolidWorks design model. If you make any changes to your SolidWorks model, all of your CAM operations are updated automatically.

Major Benefits of Using SolidCAM in SOLIDWORKS:

- SolidWorks look and feel through SolidCAM's totally seamless single window integration
- Full associativity means that tool paths automatically update when SolidWorks model changes
- SolidCAM's 10+ years as SolidWorks Gold Partner
- SolidCAM works in the SolidWorks assembly mode to define fixtures, tooling and vices
- SolidCAM + SOLIDWORKS is scalable with various packages for all your CNC-Machine types and applications
- SOLIDWORKS + SolidCAM integrated CAD/CAM solution is available from SolidCAM at a competitive bundle price

SolidCAM + SOLIDWORKS Automates IC Test System Manufacturing for Essai

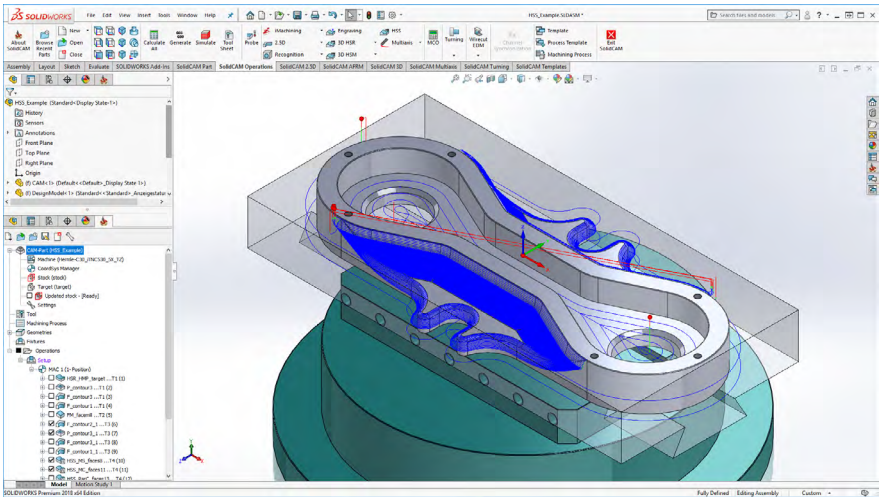
Deniz Valle, Essai Corporation Operations Manager:

- "Because SolidCAM is completely integrated with the SolidWorks CAD software, we are actually building our CAM programming within SolidWorks."
- "This approach shortens the learning curve for programmers, offers greater geometry editing and manipulation power to manufacturing, and provides a common tool for supporting interaction between designers and machinists."
- "The integrated approach facilitates discussion and resolution of manufacturing issues because everyone is working with the same model and the same modeler. We communicate issues and features a lot better working with an integrated system."

Barabi, Essai Corporation Founder & Manager:

- "If changes are made on the manufacturing side, we capture them on both the design side and the manufacturing side because SolidWorks and SolidCAM are fully associative."
- "The integrated approach has a lot of advantages, including saving time, accessing a single geometry file, and using the intelligence of our design data in a more efficient, systematic way."

You Never Have to Leave the SOLIDWORKS Window!



Larry Rehak, Intricate Metal Forming Co.:

- “In the last 45 days, since loading the SolidCAM trial version integrated in SolidWorks, I’ve been able to program complex parts and run them without concern. The machine runs smoother than before, cutters last longer and confidence levels are high. I am able to train others here to use SolidCAM with ease. The software is pretty self-explanatory and the tutorials are easy to follow.”

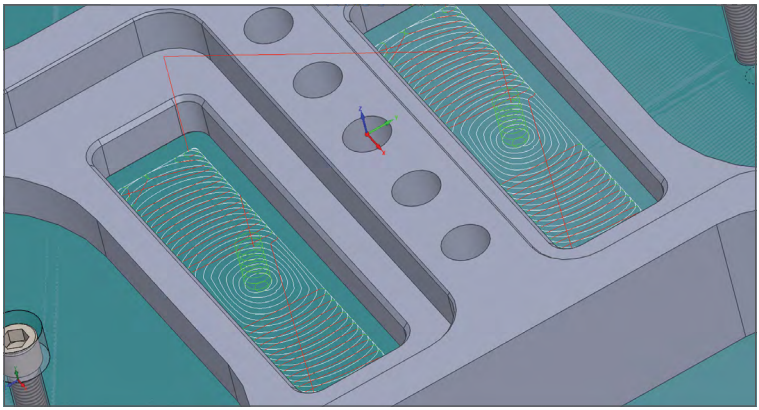
Terry Kramer, Kramer Design Corp.:

- “The tight integration with SolidWorks makes my design-to-production life cycle easy and fast. The SolidCAM support team is rock solid. I do some pretty complex 4-Axis production projects and SolidCAM handles them very nicely.”

Imagine Putting the Knowledge and Expertise of Hundreds of CAM and CNC Masters Right in the Palm of your Hand – Experience the iMachining Technology Wizard & the iMachining Tool Path!

Patented iMachining: “Simply Amazing”

This is what customers, machine tool manufacturers and tooling companies alike say about iMachining. The revolutionary iMachining CAM module, fully integrated in SolidWorks, will make you and your CNC-Machines more profitable and more competitive than ever before.



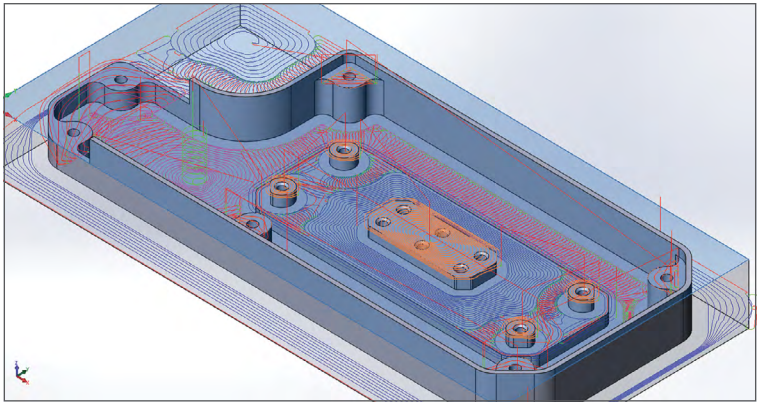
The Revolution in CNC Machining

- Saves 70% or more in CNC machining time
- Extends tool life dramatically
- Provides optimal feeds and speeds, taking into account many factors such as the tool path, stock and tool material as well as the machine specifications and limitations

iMachining provides unbelievable savings and increased efficiency in the programming of your CNC milling operations, translating into profits and success. All SolidCAM customers worldwide, who bought iMachining, are enjoying these immense savings!

Unique Technology Wizard

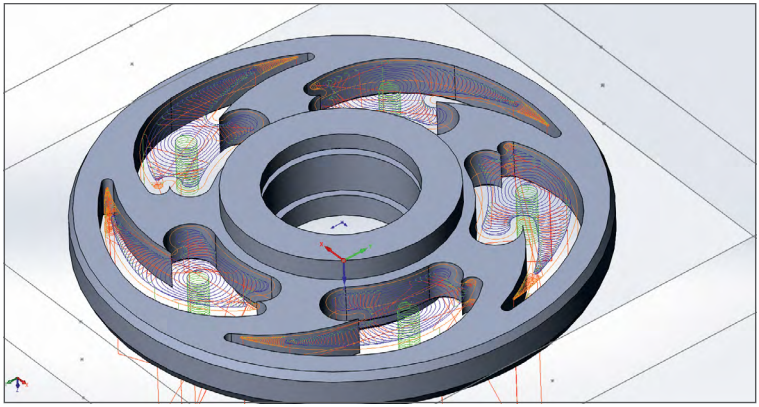
SolidCAM's iMachining has the exclusive patented iMachining Technology Wizard, the industry's first and only Wizard that automatically calculates the Cutting conditions for the iMachining tool path.



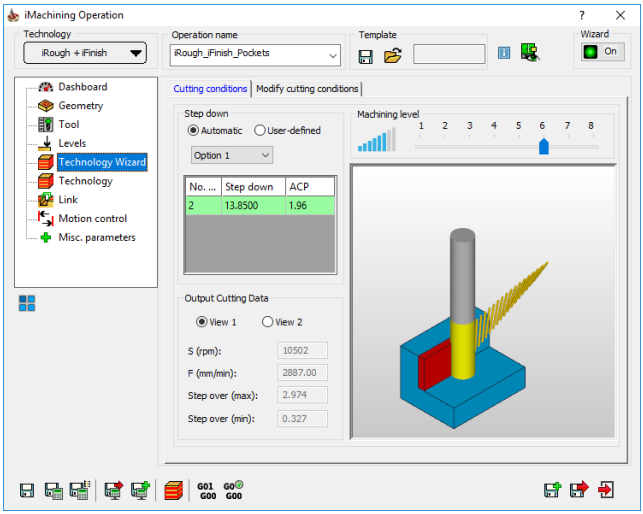
The Wizard provides optimal feeds and speeds, taking into account many factors such as the tool path, stock and tool material as well as the machine specifications and limitations.

Using "Controlled Step Over" technology, the iMachining technology strictly maintains the Cutting conditions set by the Wizard.

SolidCAM with iMachining is the only CAM system that takes out the guesswork from defining the Cutting conditions and automatically provides the optimum values for milling.



iMachining Technology Wizard + iMachining Tool Path is the Ultimate Solution!



SolidCAM's iMachining highlights:

- Increased productivity due to shorter cycles – time savings 70% and more!
- Dramatically increased tool life
- Unmatched hard materials machining
- Outstanding small tool performance
- 4-Axis and Mill-Turn iMachining
- Automatic, optimal feeds and speeds provided by the unique Technology Wizard
- High programming productivity
- Best user interface
- Shortest learning curve in the industry

"We have found all claims for iMachining to hold true in Dixon Surgical – incredible tool life, faster cycles, lighter cutting loads and protection of small cutters. The user interface is very clear and programming iMachining is faster than traditional machining strategies."

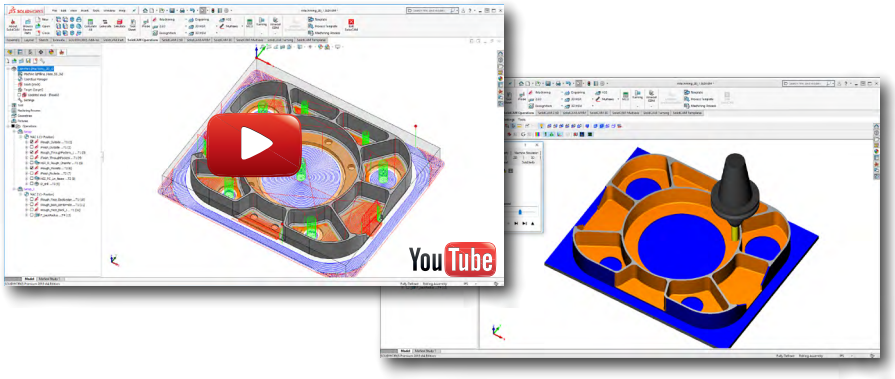
Jay Dixon, Dixons Surgical, UK

"Every day we don't use SolidCAM iMachining, we are losing money!"

Jason Near, Rotary Airlock, USA

"With iMachining, even on low-performance machines, we can reach very high Material Removal Rates."

Dreiling Maschinenbau GmbH, Germany



Click to open the SolidCAM part: **iMachining_2D_1.prz**

Click to download the recording.

The operations in this example illustrate the use of SolidCAM's iMachining 2D technology to machine the part shown above. The machining is performed on a 3-Axis CNC-Machine in two setups, using two SolidCAM Coordinate Systems.

- **Outside shape machining (iRough_Outside; iFinish_Outside)**

  These iMachining operations define the machining of the outside shape of the part. Two chains are defined, with the first being the stock boundary and the second being the profile around the part. The stock chain is marked as open, which specifies the tool should start machining from that chain. A Ø12.5 mm (0.5 in) end mill is used. iRough has a 0.25 mm (0.01 in) allowance on the walls, which is then removed by the iFinish operation.

- **Through pockets machining (iRough_ThroughPockets_1; iFinish_ThroughPockets)**

  These iMachining operations define the machining of the five circular through pockets. Five chains are defined, which represent the five through pockets. A Ø12.5 mm (0.5 in) end mill is used and since the pockets are closed with no Pre-Drilling data or entry chains defined, the tool performs helical ramping to enter into the pockets.

- **Rough machining of angled surfaces (HSR_R_Rough_Chamfer)**

  This HSR operation defines the rough machining of the four large chamfers on the ribs of the part. Two boundaries are picked off the edges the make up the chamfers. A Ø12.5 mm (0.5 in) end mill is used with the Tool Relation set to Centered. A 1.27 mm (0.05 in) Step down is specified with a 0.13 mm (0.005 in) allowance to remain on the surfaces.

- **Pockets machining (iRough_Pockets; iFinish_Pockets)**

  These iMachining operations define the machining of the three semi-open pockets and the seven closed pockets. Since all ten pockets have the same depth, they can all be machined in one operation. A Ø10 mm (0.4063 in) bull nose mill with a corner radius of 1.6 mm (0.0625 in) is used. iRough has a 0.25 mm (0.01 in) allowance on the walls, which is then removed by the iFinish operation.

- **Finish machining of angled surfaces (HSS_PC_Lin_faces)**

  This HSS operation defines the finish machining of the four large chamfers on the ribs of the part. The Ø10 mm (0.4063 in) bull nose mill is used and the Linear strategy generates Parallel cuts with a 0.5 mm (0.02 in) Maximum step over. Customized linking is used to allow short repositions and smooth transitions when starting each cut.

- **Holes machining (D_drill)**

  This Drilling operation defines the drilling of the holes through the lower surface of the part.

- **Bottom ledge machining (iRough_Face_BackLedge)**

  This iMachining operation defines the bottom ledge machining on the underside of the part. Two chains are defined, with the first being the stock boundary and the second being the bottom of the floor radius. Using a Ø12.5 mm (0.5 in) end mill, the tool starts machining from the stock chain and collapses in towards the radius.

- **Cutting excess material from through hole (iRough_back_centerHole)**

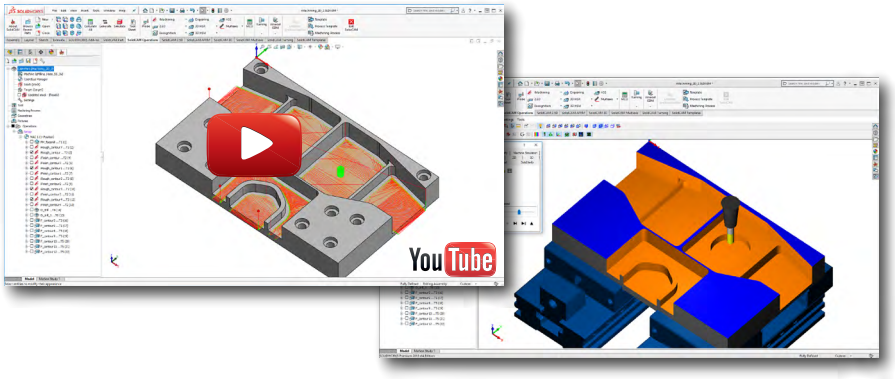
  This iMachining operation is defined to machine away the excess material from the center through hole of the part. This excess material was used for clamping in the first setup.

- **Bottom face machining (iRough_Face_Back_1)**

  This iMachining operation defines the bottom face machining on the underside of the part. Using a Ø12.5 mm (0.5 in) end mill, a morphing spiral tool path is performed from the outer open chain, collapsing towards the inner closed chain.

- **Floor radius finishing (F_backRadius)**

  This Profile operation defines the 6.35 mm (0.25 in) floor radius finishing on the underside of the part. The chain is defined as the bottom edge of the radius. A Ø12.5 mm (0.5 in) ball nose mill is used with the Tool side set to Center. A 0.13 mm (0.005 in) offset remains on the floor after the first roughing pass and is then removed with a single finishing pass. A 0.25 mm (0.01 in) arc is used for the Lead in/out.



Click to open the SolidCAM part: **iMachining_2D_2.prz**

Click to download the recording.

The operations in this example illustrate the use of SolidCAM's iMachining 2D technology to machine the part shown above. The machining is performed on a 3-Axis CNC-Machine in one setup, using one SolidCAM Coordinate System.

- **Upper face machining (FM_facemill)**

  This Face Milling operation defines the machining of the part upper face. A Ø100 mm (4 in) face mill is used with the Hatch technology to remove 1 mm (0.04 in) of material from the stock top face.

- **Pocket machining (iRough_contour7; iRough_contour; iFinish_contour; iFinish_contour7; iRough_contour1; iFinish_contour1)**

  These iMachining operations define the machining of the semi-open pocket. A Ø12 mm (0.4844 in) end mill is used. The iRough operations have a 0.25 mm (0.01 in) allowance on the walls, which is then removed by the iFinish operations.

- **Pockets machining (iRough_contour2; iFinish_contour2; iRough_contour3; iFinish_contour3; iRough_contour4; iFinish_contour4)**

  These iMachining operations define the machining of the semi-open pockets and the two closed pockets. The first iRough and iFinish operation machines the two larger semi-open pockets. The second and third pair of operations machine the smaller semi-open and closed pockets. A Ø12 mm (0.4844 in) end mill is used. The iRough operations have a 0.25 mm (0.01 in) allowance on the walls, which is then removed by the iFinish operations.

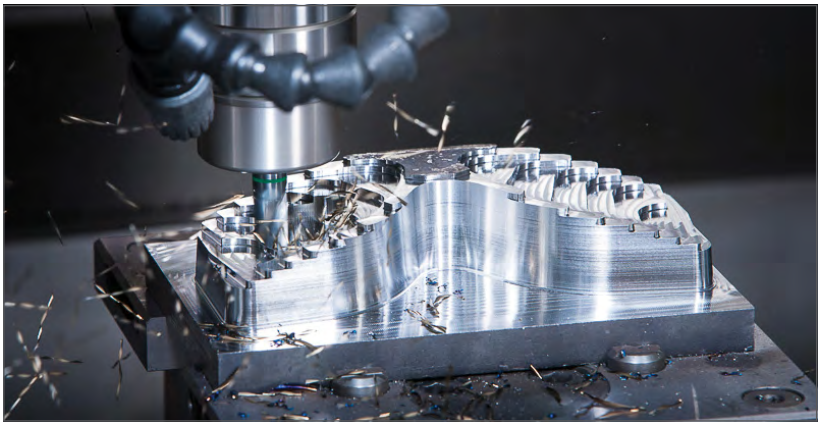
- **Holes machining (D_drill; D_drill_1; F_contour5)**

  These Drilling operations define the center drilling and drilling of the holes located on the upper surface of the part. The Profile operation is defined to finish the walls of the holes.

- **Chamfer machining (F_contour6 – F_contour12)**

  These Profile operations define the machining of the chamfers on the part edges using a Ø6 mm (0.2344 in) chamfer drill and the Chamfer option.

Uses Proven iMachining 2D & Technology Wizard Algorithms for Roughing, Rest Machining and Semi-Finishing of Molds, Complex 3D Parts and 3D Prismatic Parts



iMachining 3D provides amazing 3D machining results, regularly saving 70% and more in machining time, even reaching up to 90% savings.

iMachining 3D produces a complete, ready to run CNC program, with optimal cutting conditions achieved by the expert knowledge-based Technology Wizard, to rough and rest rough a entire 3D part, all in a single operation.

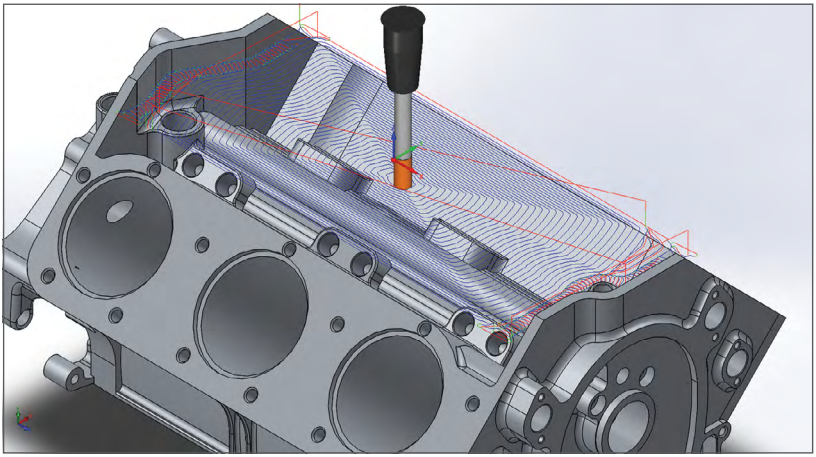
All the volumes requiring removal are subdivided into milling regions. By default, sophisticated analysis algorithms are then used to determine the optimal order of Step down (roughing) and Step-up (rest roughing) tool path passes. By achieving its unique local machining feature, iMachining 3D is able to eliminate almost all retracts and long position moves.

With its full-depth Step down, intelligent Step-up and its smart positioning, almost all retracts, long positioning and air cutting are eliminated to achieve the shortest optimal cycle times in the industry for the roughing, rest roughing and semi-finishing of molds, complex 3D parts and 3D prismatic parts.

iMachining 3D provides a complete machining solution for your 3D parts when combined with other SolidCAM Milling technologies for finishing.

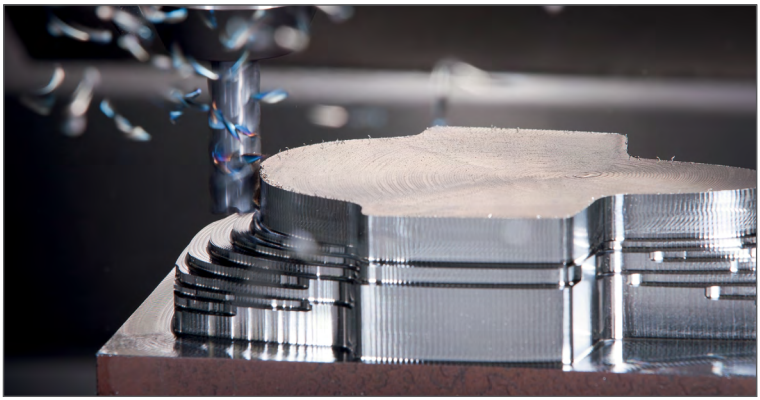
Exclusive iMachining 3D Features:

- Quick solid model geometry selection, eliminating the need to define chain geometries
- Optimized machining of each Z-level, using proven iMachining 2D tool path
- Deep roughing utilizes the whole length of the flute, shortening cycle time and increasing tool life
- Rest material machining in small upward steps, optimized for constant scallop height, further shortens cycle time
- Intelligent localized machining and optimal ordering eliminates almost all retracts and long positioning moves, producing the shortest cycle times in the industry
- A dynamically updated 3D Stock model eliminates all air cutting
- Tool path automatically adjusts to avoid contact between the holder and updated stock at every stage of the machining process



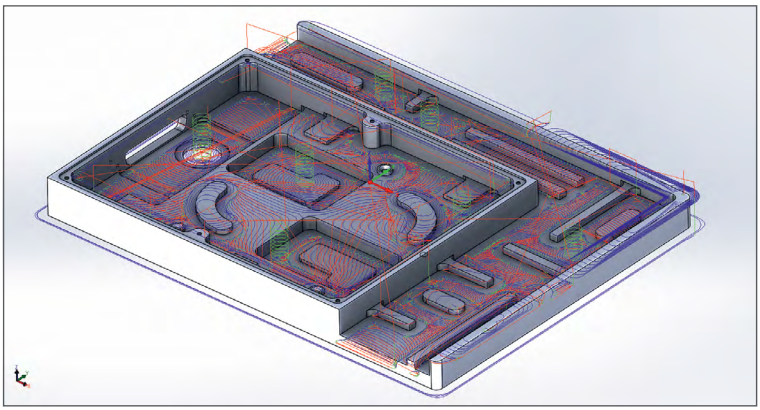
iMachining 3D for General Shaped Parts

iMachining 3D provides a complete machining solution for your general shaped 3D parts when combined with 3D HSM for finishing molds and complex 3D parts.



iMachining 3D for Prismatic Parts

You can also completely rough your 3D prismatic parts that include multiple pockets and islands on different Z-levels, all in a single operation. In this case, iMachining 3D provides a complete machining solution when combined with the 2.5D Milling and AFRM modules for finishing your 3D prismatic parts. You can also use iMachining 3D to rough your 3D prismatic parts entirely and then use iMachining 2D to finish the floors and walls.



iMachining 3D Saves Menes 85% in Cycle Time with Steel:

"I hope we get such results every day – we will have great savings in costs with iMachining 3D!"

Igor, Chief Programmer, Menes

iMachining 3D Cuts Machining Time on a Makino by 75% Against Another CAM System:

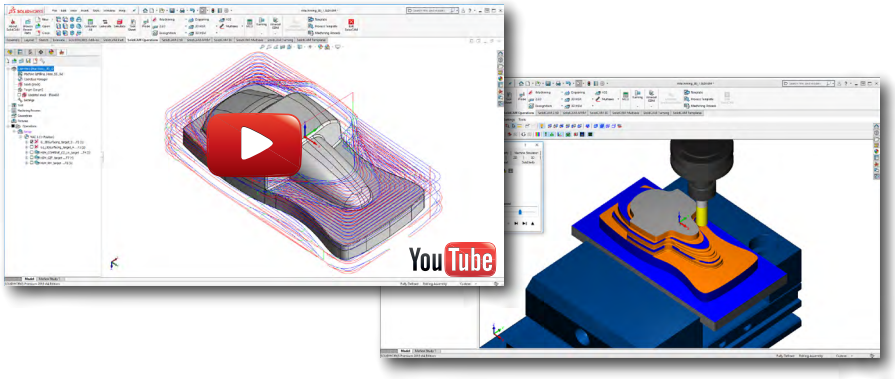
"iMachining 3D cut machining time from 4 hours, with a competitor's CAM system, down to 58 minutes ... a 75% time savings!"

Galtronics, China

SolidCAM Customer A.P.A. on iMachining 2D & 3D, Mainly for Aluminum Machining:

"Amazing, I have no words to describe my satisfaction with iMachining. I can't even express how much time it saves us in Aluminum!"

David Franko, Owner, A.P.A.



Click to open the SolidCAM part: **iMachining_3D_1.prz**

Click to download the recording.

The operations in this example illustrate the use of SolidCAM's iMachining 3D technology to machine the part shown above.

- **Roughing (G_i3DSurfacing_target_3)**

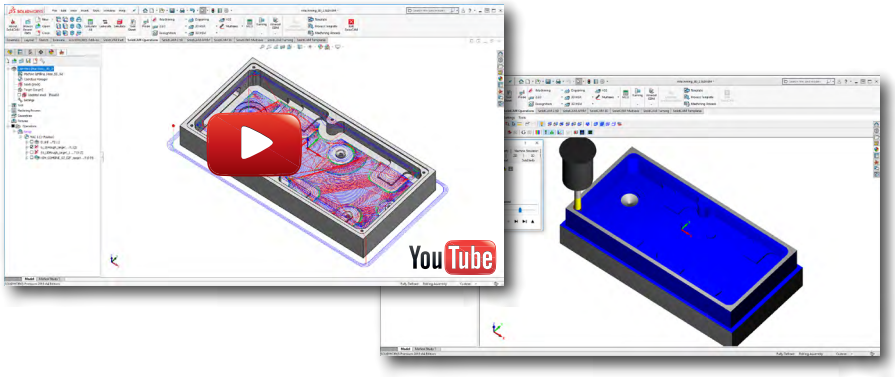
  This iMachining 3D operation is defined to remove the majority of material using the 3D General strategy and a Ø16 mm (0.625 in) end mill. After 4 Step downs, rest roughing is performed in Step-up mode with the current tool to remove rest material on sloped surfaces according to the automatically defined value of Scallop. A machining allowance of 0.2 mm (0.008 in) remains on the surfaces.

- **Rest machining (G1_i3DSurfacing_target_4)**

  This iMachining 3D operation defines the removal of unmachined material remaining after the previous operation and uses a Ø12 mm (0.4844 in) bull nose mill with a corner radius of 0.6 mm (0.024 in). After 2 Step downs, rest roughing is performed in Step-up mode with the current tool to remove additional rest material on sloped surfaces according to the automatically defined value of Scallop. A machining allowance of 0.2 mm (0.008 in) remains on the surfaces.

- **Finish machining (HSM_COMBINE_CZ_Lin_target; HSM_CZF_target; HSM_RM_target)**

  These HSM operations use the Combine, Horizontal and Rest machining strategies to define the part finishing.



Click to open the SolidCAM part: **iMachining_3D_2.prz**

Click to download the recording.

The operations in this example illustrate the use of SolidCAM's iMachining 3D technology to machine the part shown above.

- **Through hole machining (D_drill)**

  This Drilling operation defines the drilling of the through hole. The hole will be used for safe tool entry in the subsequent iMachining operation. A Ø16 mm (0.625 in) drill is used.

- **Roughing (G_i3DRough_target)**

  This iMachining 3D operation is defined to remove the majority of material using the 3D General strategy and a Ø12 mm (0.4844 in) end mill. The 3D Prismatic strategy can also be used in this case. After 3 Step downs, rest roughing is performed in Step-up mode with the current tool to remove rest material on higher horizontal surfaces. A machining allowance of 0.38 mm (0.015 in) remains on the wall and floor surfaces.

- **Rest machining (G1_i3DRough_target_1)**

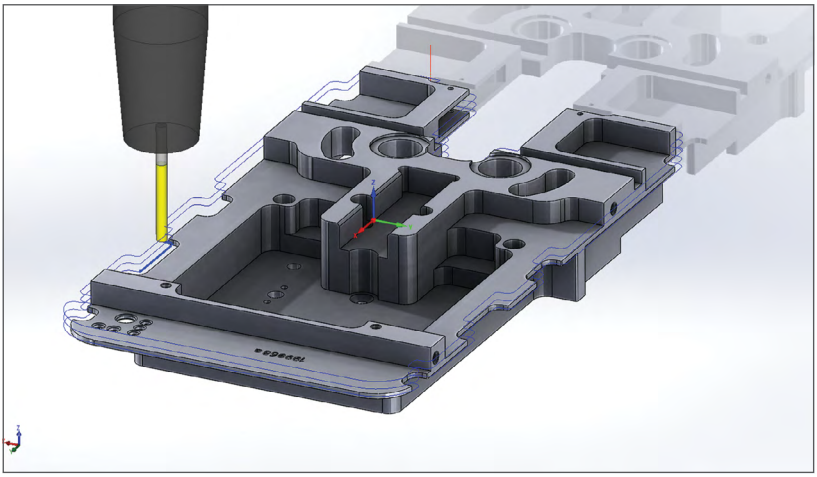
  This iMachining 3D operation defines the removal of unmachined material remaining in the corners after the previous operation and uses a Ø6 mm (0.2344 in) end mill. A machining allowance of 0.38 mm (0.015 in) remains on the wall and floor surfaces.

- **Finish machining (HSM_COMBINE_CZ_CZF_target)**

  This HSM operation combines the Constant Z and Horizontal strategies to define the part finishing.

The Most Powerful and Easiest to Create 2.5D CNC Milling Tool Paths: Interactive Control + Feature Recognition!

The most straightforward, easy-to-use interface, seamlessly integrated in SolidWorks, combined with the latest tool path technology, provides the fastest, most powerful and easiest to create 2.5D CNC Milling tool paths.



Easily work on parts, assemblies and sketch geometry to define your machining operations. Quickly place fixtures and components for full visualization.

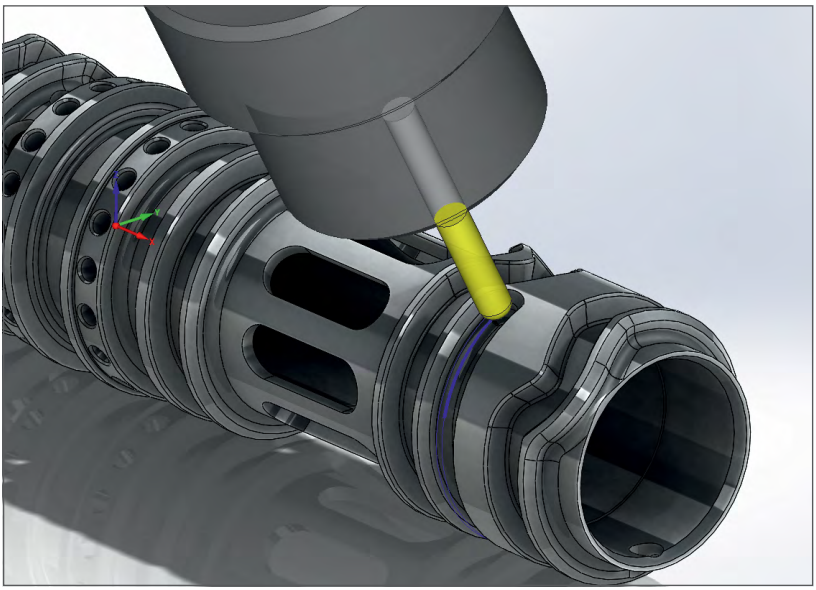
Best of Both Worlds: Complete Interactive Control + Feature Recognition

SolidCAM provides both interactive and automated 2.5D Milling operations on SolidWorks models. Designed for both the novice and advanced user, SolidCAM offers the best of both worlds, with your choice of fully controlled selection of geometry, parameters and CNC programming strategies or with the Automatic Feature Recognition and Machining module for Pocket, Chamfer and Drill geometries.

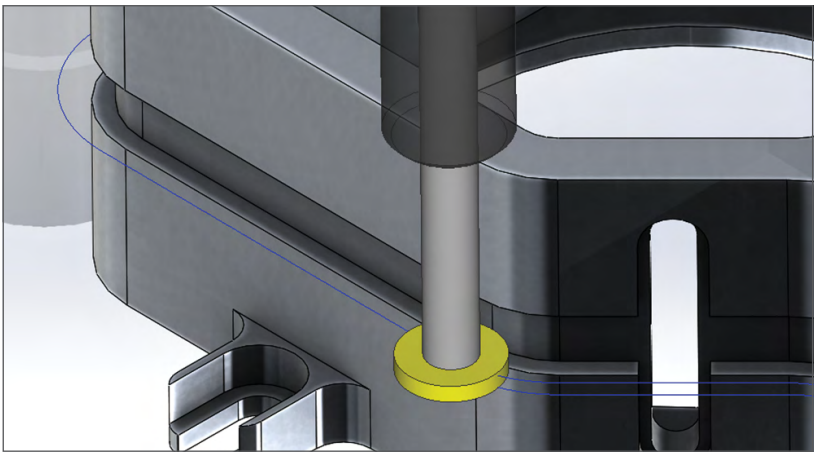
Interactive 2.5D Milling Operations

Aside from the standard 2.5D Milling strategies such as Profile, Pocket and Drilling operations, SolidCAM offers:

- Chain modification options (offsetting, trimming, extending etc.), enabling changes to geometry without changing the CAD model
- Automatic rest material machining to cut the material that remains after using larger tools
- Chamfer machining using the same geometry defined in Profile and Pocket operations
- Thread Milling operation for machining of standard internal and external threads
- Engraving of text on flat and wrapped faces and middle line engraving of multi-line text
- Contour 3D operation drives the tool along a 3D curve, cutting the model at different depths
- Machining of geometry wrapped around rotation axes, by transforming linear movements to rotary movements

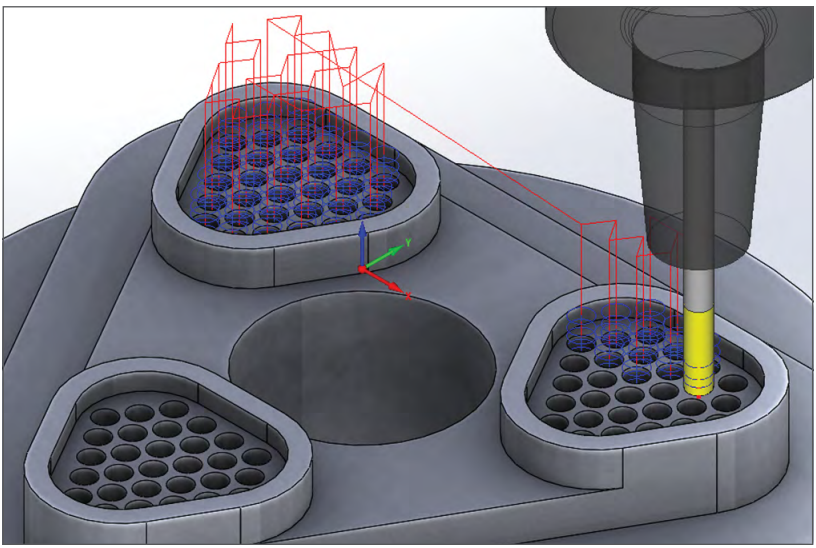


- Special operation for machining of side slots with undercut using a T-slot tool



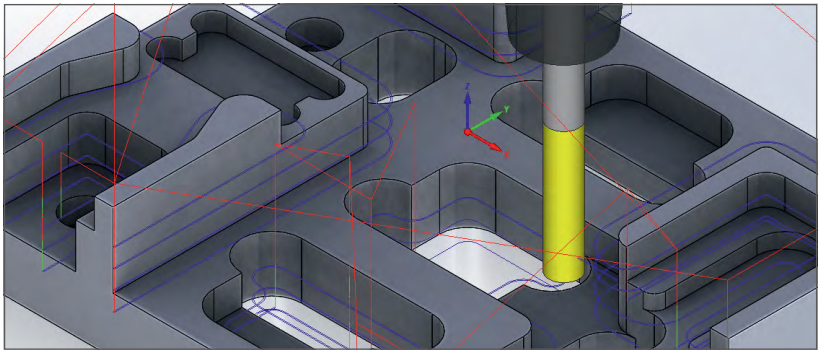
Drill Recognition

Automatic recognition and grouping of holes from the CAD model with the option to modify resulting geometry. A single Drill Recognition operation can machine groups of holes that have various levels and cutting depths.



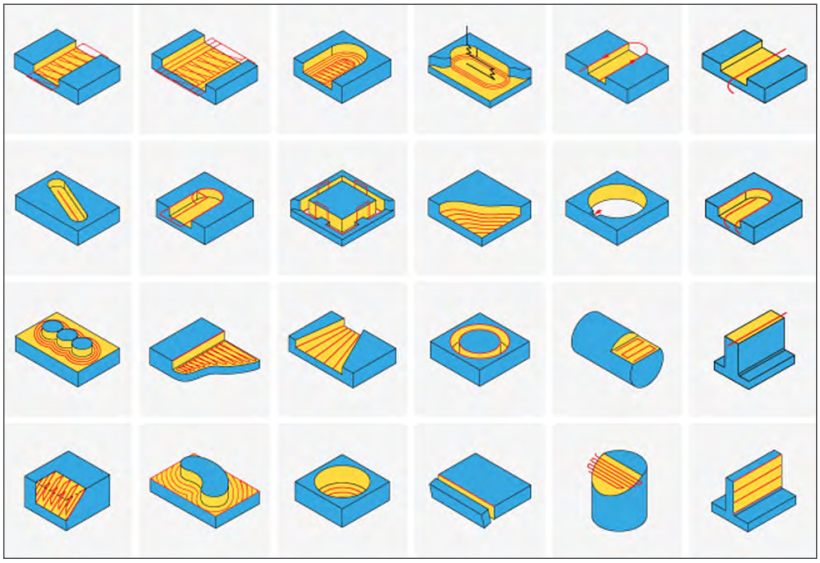
Pocket Recognition

SolidCAM's powerful pocketing strategy is taken to the next level with the automatic recognition of all pockets from the CAD model. All strategies and options of the standard Pocket operation are available, with the capability to machine pockets that have various levels and cutting depths.

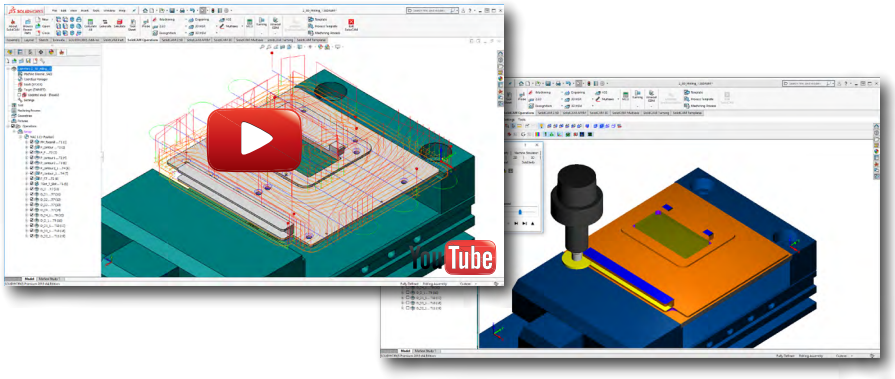


Toolbox Cycles

The useful and convenient Toolbox Cycles provide specialized sub-operations for slots, corners, bosses, ruled surfaces, etc.



2.5D Milling



Click to open the SolidCAM part: **2_5D_Milling_1.prz**

Click to download the recording.

The operations in this example illustrate the use of several SolidCAM 2.5D Milling technologies to machine the part shown above. The machining is performed on a 5-Axis CNC-Machine in one setup, using one SolidCAM Coordinate System.

- **Upper face machining (FM_facemill)**

  This Face Milling operation defines the machining of the part upper face. A Ø40 mm (1.6 in) face mill is used with the Hatch technology to remove 0.5 mm (0.02 in) of material from the stock top face.

- **Outside shape machining (F_contour)**

  This Profile operation defines the machining of the outside shape. The material is removed in three step down cuts using a Ø16 mm (0.625 in) end mill, and an additional finish pass is executed to remove the remaining 0.3 mm (0.012 in) from the wall.

- **Island and pockets machining (P_F; P_contour1; P_contour2)**

  These Pocket operations define the machining of the island and pockets. The first operation creates the large island using a Ø16 mm (0.625 in) end mill. The Open pocket technology uses a profile strategy and approaches from the outside, leaving a 0.2 mm (0.008 in) allowance on the walls of the island for a finishing pass. In the second operation, the same tool is used to create nubs on the island upper face. In the third operation, the through pocket is created inside the island using a Ø10 mm (0.4063 in) end mill.

- **Corners rest and finish machining (P_contour2_1; F_contour_1)**

  These operations define the inside corners machining of the through pocket and outside shape. Both the Pocket operation and the Profile operation use a Ø3 mm (0.125 in) end mill with the Rest material option to only remove the material that remains after the larger tool from the previous operations.

- **Chamfer machining (F_F7)**

  This Profile operation defines the machining of the chamfers on the part edges using an Ø8 mm (0.3125 in) chamfer drill and the Chamfer option.

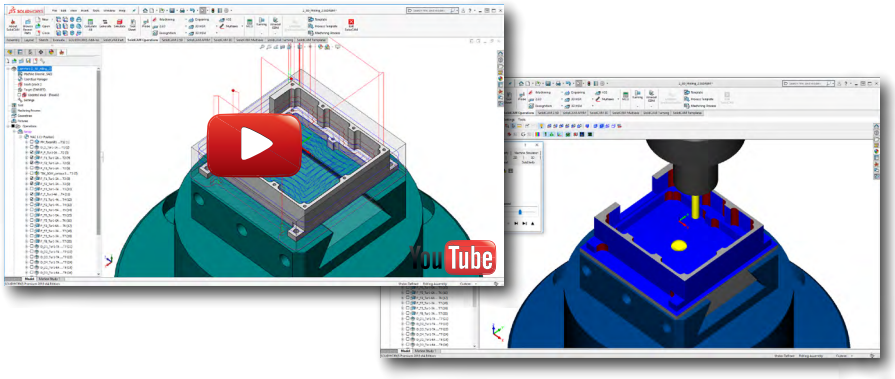
- **Slot machining (TSlot_T_Slot)**

  This T-Slot operation is defined to machine the side groove. A Ø36 mm (1.4 in) slot mill is used and finish passes are executed on the groove floor, ceiling and walls.

- **Holes machining (D_D – D_D2_1)**

  These Drilling operations define the center drilling and drilling of the holes located on all the part faces.

2.5D Milling



Click to open the SolidCAM part: **2_5D_Milling_2.prz**

Click to download the recording.

The operations in this example illustrate the use of several SolidCAM 2.5D Milling technologies to machine the part shown above. The machining is performed on a 5-Axis CNC-Machine in two setups, using two SolidCAM Coordinate Systems.

- **Upper face machining (FM_facemill1)**

  This Face Milling operation is defined to machine the upper face of the part in only one pass using a Ø100 mm (4 in) face mill.

- **Pockets machining (D_D_Tur1-1A; P_F2_Tur1-2A; P_F3_Tur1-2A)**

  These operations define the rough machining of the center pocket areas. In the Drilling operation, a hole is drilled using a Ø10 mm (0.4083 in) drill that will be used for safe tool entry in the subsequent Pocket operations. A Ø10 mm (0.4063 in) rough mill is then used with the Contour strategy to perform the pocket machining.

- **Outside shape machining (F_F_Tur1-2A; F_F1_Tur1-2A)**

  These Profile operations define the rough machining of the outside shape. A Ø10 mm (0.4063 in) rough mill is used, leaving a 0.2 mm (0.008 in) allowance on the walls.

- **Slot machining (TBX_SOW_contour3)**

  This ToolBox cycles operation is defined to machine a wide open slot in the part side wall. A spiral tool path effectively removes the material using the whole cutting length of the Ø10 mm (0.4063 in) rough mill.

- **Corners rest and finish machining (F_F1_Tur1-3A; F_F1_Tur1-5A)**

  These Profile operations define the inside corners machining of the external tabs. Both operations use a Ø5 mm (0.2031 in) rough mill with the Rest material option to only remove the material that remains after the larger tool from the previous operations. In the second operation, the tool finishes the corners at the full depth.

- **Semi-finish and finish machining (F_F_Tur1-4A; F_F1_Tur1-4A; F_F2_Tur1-3A; F_F3_Tur1-3A; F_F2_Tur1-4A; F_F6_Tur1-5A; F_F5_Tur1-6A; F_F3_Tur1-6A)**

  These Profile operations define the semi-finish and finish machining of the part external and internal surfaces using different end mills. A smaller end mill is used for the finishing.

- **Chamfer machining (F_F7_Tur1-7A; F_F3_Tur1-7A; F_F8_Tur1-7A)**

  These Profile operations define the machining of the chamfers on the part edges using a Ø2 mm (0.0781 in) chamfer drill.

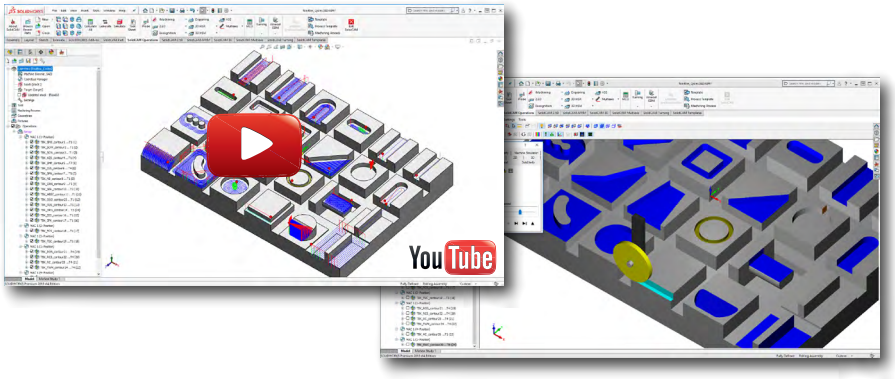
- **Holes machining (D_D1_Tur1-7A – D_D4_Tur1-11A)**

  These Drilling operations define the center drilling and drilling of the holes located on all the part faces.

- **Face and chamfer machining on the part lower surfaces (FM_facemill2; F_contour4)**

  In the second setup, the Face Milling operation defines the machining of the part bottom surface and the Profile operation is defined to machine away the excess material around the edges on the lower side of the part.

2.5D Milling – ToolBox Cycles



Click to open the SolidCAM part: **ToolBox_Cycles.prz**

Click to download the recording.

The operations in this example illustrate the use of SolidCAM's ToolBox Cycles to machine the part shown above. The machining is performed on a 5-Axis CNC-Machine in one setup, using one SolidCAM Coordinate System with multiple positions. The following operations are included but not limited to:

- **Closed slot machining (TBX_CLS_contour6)**

  This cycle enables you to machine a closed slot with the width equal to the tool diameter. The machining is performed in the zigzag manner; the last cut is horizontal to clean the slot floor.

- **Spine slot machining (TBX_SPN_contour7)**

  In this cycle, SolidCAM automatically determines the spine of the slot and performs the machining according to the spine shape.

- **Four nubs machining (TBX_NB_contour8)**

  This cycle enables you to machine the profile of a pocket while preserving the rest of the material fixed to the wall using small thin bridges.

- **Simple corner machining (TBX_CRN_contour9)**

  This cycle enables you to machine an open corner area with a number of cuts equidistant to the selected geometry.

- **Simple multi-bosses machining (TBX_MBSC_contour11)**

  This cycle enables you to machine several bosses starting from one face. The machining is performed in a number of equidistant Clear offset passes parallel to the selected bosses geometry.

- **One side open slot machining (TBX_OSO_contour25)**

  This cycle enables you to machine a one-sided slot using two chains: a main chain and a drive chain. The machining is performed with a spiral tool path.

- **O-Ring machining (TBX_ORG_contour14)**

  This cycle enables you to machine slots that have an O-ring shape. The geometry is defined as a pair of closed chains, with the first being an external chain and the second being an internal chain.

- **Spiral pocket machining (TBX_SPK_contour17)**

  This cycle enables you to create a spiral-shaped tool path inside the pocket. The Spiral cuts are controlled by Cutting angle parameters, ranging from minimal to maximal values.

- **Roll into open slot (TBX_ROS_contour21)**

  This cycle enables you to machine an open slot that, while rolling into the material with an arc movement, starts from the slot center and diverges to the walls.

- **Roll into closed slot (TBX_RCS_contour22)**

  This cycle enables you to machine a closed slot that, while rolling into the material with an arc movement, starts from the slot center and diverges to the walls.

- **Rib cut (TBX_RC_contour23)**

  This cycle enables you to machine a rib upper surface in a single pass shifted from the center line by a specified value. This is a safe method of cutting the upper face of a thin wall.

- **Thin wall machining (TBX_TWM_contour24)**

  This cycle enables you to machine thin walls by gradually removing the excess material from either side of the wall. Alternating steps and Supporting steps strategies are supported.

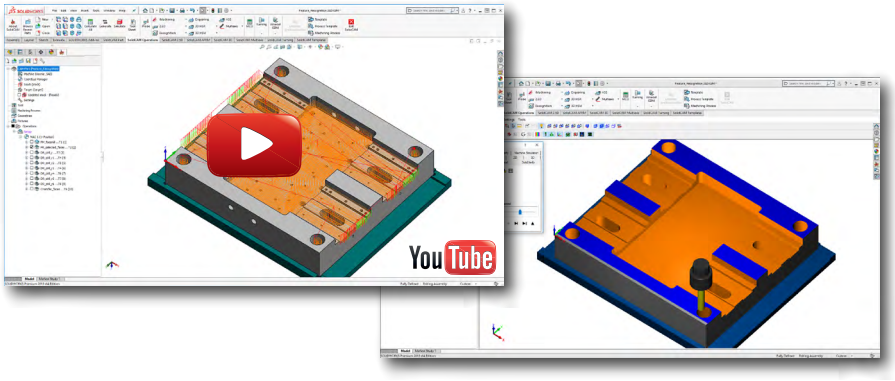
- **Angled cylinder machining (TBX_AC_contour26)**

  This cycle enables you to create an angled surface on the top of a cylinder. The cutting is performed by parallel cuts with a defined Step over.

- **Wood cutting (TBX_SWC_contour20)**

  This cycle enables you to use a special Saw tool to cut off a block of wood. The First step down parameter defines the depth of cutting during the first pass. The second pass completes the cutting at the full depth.

2.5D Milling – Feature Recognition



Click to open the SolidCAM part: **Feature_Recognition.prz**

Click to download the recording.

The operations in this example illustrate the use of SolidCAM's Automatic Feature Recognition and Machining technology to machine the part shown above. The machining is performed on a 5-Axis CNC-Machine in one setup, using one SolidCAM Coordinate System.

- **Upper face machining (FM_facemill)**

  This Face Milling operation defines the machining of the part upper face. A Ø38 mm (1.5 in) face mill is used with the Hatch technology to remove material from the stock top face.

- **Pockets machining (PR_selected_faces)**

  This Pocket Recognition operation automatically recognizes and defines the machining of all the part pocket features. Using a Ø20 mm (0.75 in) end mill, the Open pocket technology performs the approach movement from an automatically calculated point outside the material. The tool descends to the necessary depth and then moves horizontally into the material. A special strategy is applied to the through pockets, which require deepening in order for them to be machined completely.

- **Center drilling (DR_drill_r)**

  This Drill Recognition operation automatically recognizes and defines the center drilling of all the part hole features. A Ø10 mm (0.4083 in) spot drill is used and the chamfer depth is applied automatically for each group of holes.

- **Drilling (DR_drill_r1 – DR_drill_r6)**

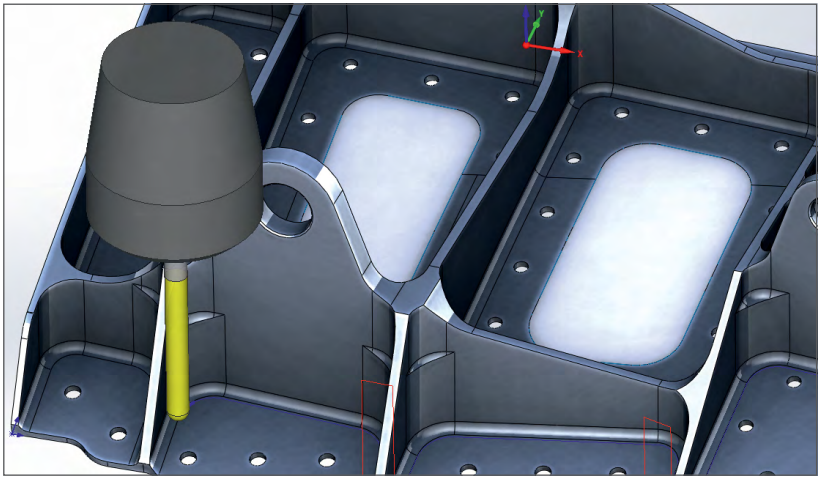
  These Drill Recognition operations define the drilling of all the part hole features. For each operation, SolidCAM automatically recognizes the hole features according to specific filters. The drilling depth is determined automatically and the through holes are extended in order for them to be machined completely.

- **Chamfer machining (CHamfer_faces)**

  This Chamfer Recognition operation automatically recognizes and defines the machining of all the part chamfer features. A Ø10 mm (0.4063 in) chamfer mill is used to machine a 0.5 mm (0.02 in) Chamfer depth. The tool descends and creates chamfers on the part edges with a Cutting diameter of 2 mm (0.08 in), while maintaining a 4 mm (0.16 in) Safety offset.

High Speed Surface Machining of Localized Surfaces is an Important Module for Every Machine Shop!

SolidCAM HSS is the High Speed Surface Machining module for smooth and powerful machining of localized surface areas, which even includes undercuts. It provides easy selection of the surfaces to be machined, with no need to define boundaries. It supports both standard and shaped tools.



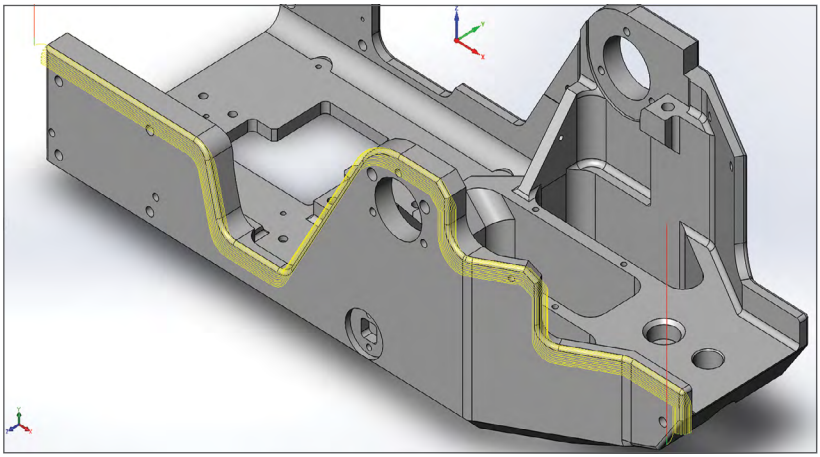
Powerful Surface Machining Strategies for Smooth, Gouge-Free and Optimal Tool Paths

The SolidCAM HSS module provides numerous surface machining strategies, that produce an efficient, smooth, gouge-free and optimal tool path to finish the selected surfaces.

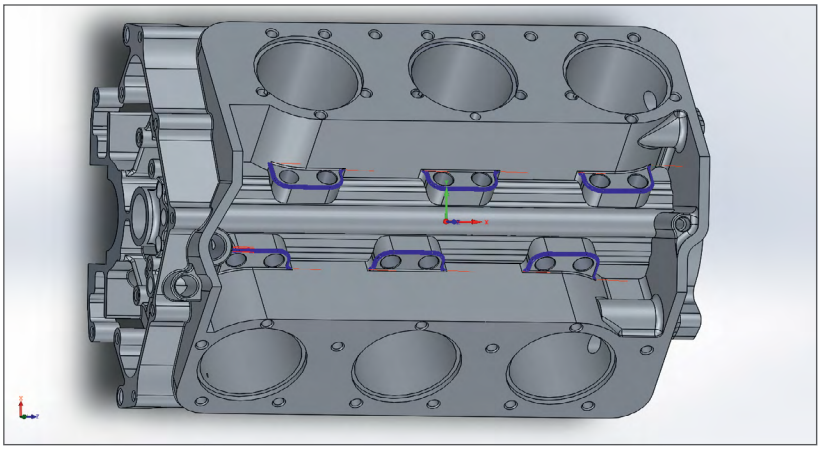
HSS provides special tool path linking options, generating smooth and tangential lead-ins and lead-outs. The linking moves between the tool paths can be controlled by the user to avoid holes and slots, without the need for modifying the surface of the model. Retracts can also be performed to any major plane.

Total Tool Control to Machine Only the Areas You Choose

HSS is the CAM module that takes your 2.5D Milling way beyond profiles, pockets and faces, providing a 3D machining capability by driving along specific surfaces on both general shaped 3D parts and 3D prismatic parts.



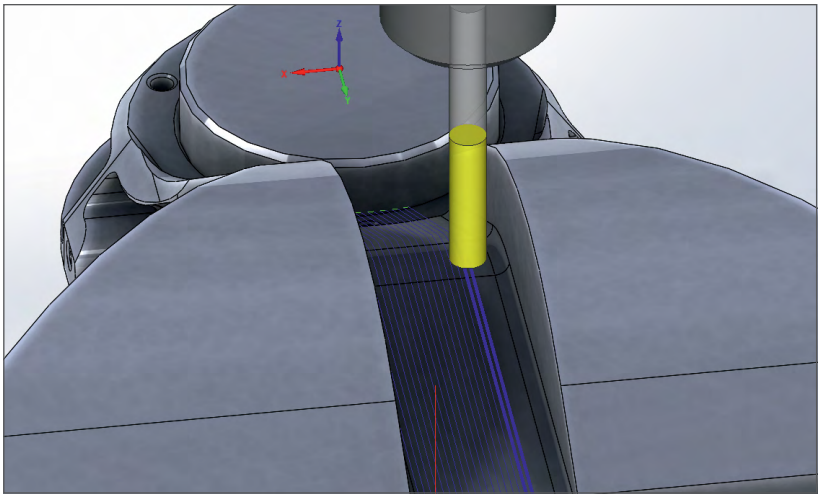
The HSS tool path is focused on single or multiple surfaces and excels in creating a flowing tool path on a group of surfaces that make up a complex 3D shape such as a fillet.



Experience total tool control to machine only the areas you choose, without the need of constraint boundaries or construction geometry.

Advanced Gouge Control for Holder, Arbor and Tool

Complete gouge control is available for holders, arbors and tools. Adjoining check surfaces that are to be avoided can be selected. Several retract strategies are available with full user control.



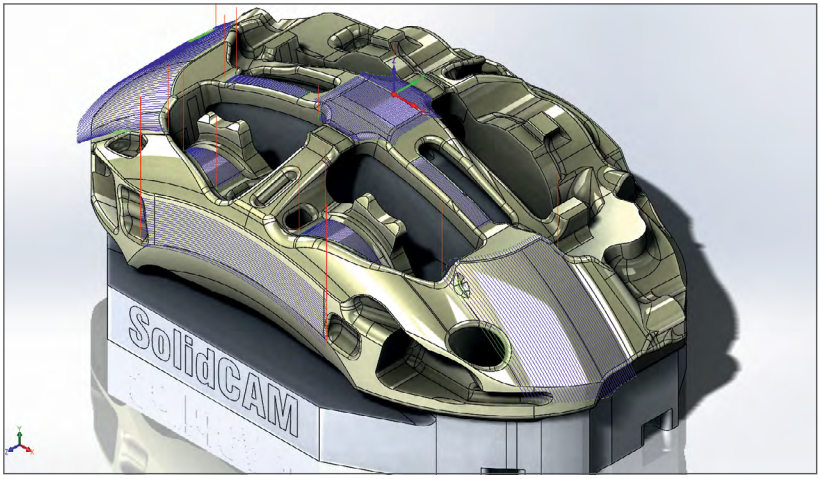
Advanced Linking

Total freedom to control tool entry and tool exit motion for avoiding surface modifications. Tool paths can be extended or trimmed, gaps and holes can be jumped, and you can choose from multiple lead-in and lead-out options.

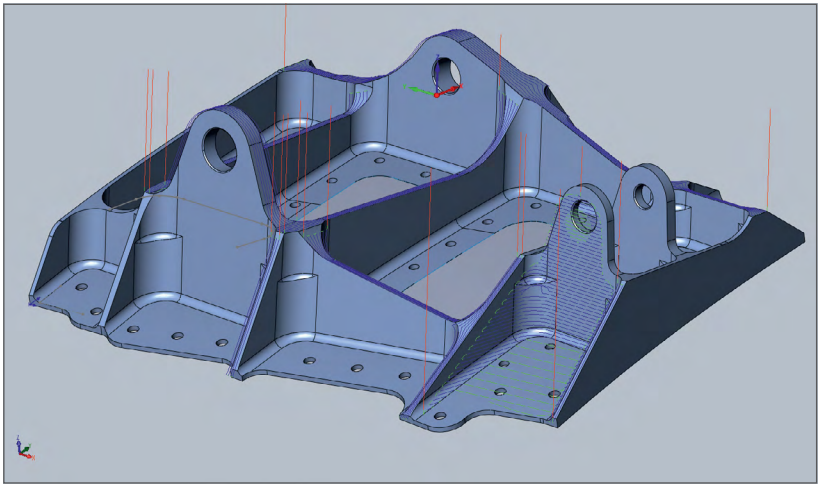
Handling Undercuts in HSS

Use Tapered, Lollipop, or T-Slot tools for undercuts or other types of geometry that are difficult to machine.

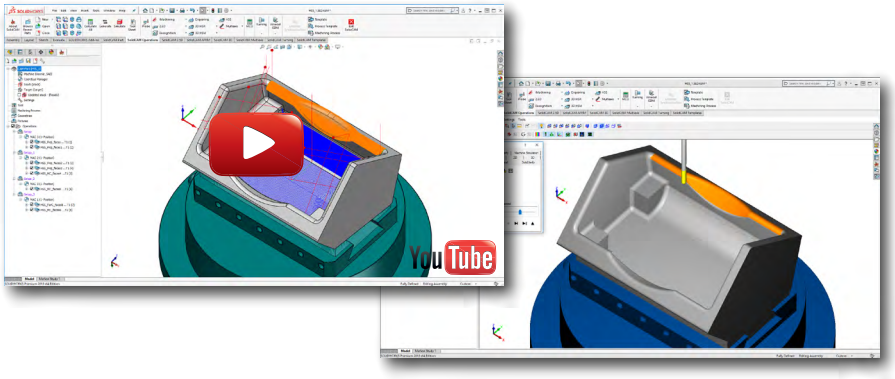
Important Module for Every Machine Shop



The advantages of the SolidCAM HSS module translates to significantly increased surface quality. It is a critical add-on for every machine shop for the machining of all types of parts.



High Speed Surface Machining (HSS)



Click to open the SolidCAM part: **HSS_1.prz**

Click to download the recording.

The operations in this example illustrate the use of several SolidCAM High Speed Surface Machining (HSS) strategies to machine the part shown above.

- **Projection machining (HSS_Proj_faces; HSS_Proj_faces1; HSS_Proj_faces2; HSS_Proj_faces3)**

  These HSS operations use the Projection strategy to define the machining of four small internal fillet areas. A Ø10 mm (0.4063 in) ball nose mill is used to fit the corresponding surface radius. For Roughing and More, the Depth cuts option ensures the whole depth is machined in several cutting passes.

- **Morph between curves machining (HSS_MC_faces4; HSS_MC_faces6)**

  These HSS operations use the Morph between two boundary curves strategy to define the machining of two large internal fillet areas. This strategy generates a constant tool path that is distributed evenly between the fillet boundaries. The Ø10 mm (0.4063 in) ball nose mill is used. Gouge check is enabled to automatically detect and avoid possible collisions between the tool and the faces of the machining area.

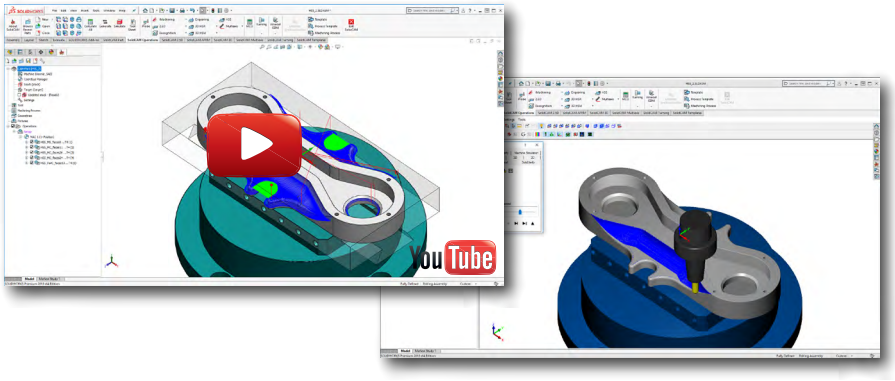
- **Parallel to curves machining (HSS_ParC_faces8)**

  This HSS operation uses the Parallel to curves strategy to define the machining of the part lower face. With this strategy, faces can be machined with cutting passes parallel to the selected curve. In this case, a pocket-style tool path is generated within the boundaries of the selected face using a Ø10 mm (0.4063 in) end mill.

- **Morph between curves machining (HSS_MC_faces9)**

  This HSS operation uses the Morph between two boundary curves strategy to define the machining of the external fillet and an inclined face adjacent to the fillet. Using a Ø6 mm (0.2344 in) ball nose mill, the tool path is generated according to a 0.004 mm (0.0002 in) Scallop value in order to obtain excellent surface quality. Gouge check is enabled to automatically detect and avoid possible collisions between the tool and the faces of the machining area.

High Speed Surface Machining (HSS)



Click to open the SolidCAM part: **HSS_2.prz**

Click to download the recording.

The operations in this example illustrate the use of several SolidCAM High Speed Surface Machining (HSS) strategies to machine the part shown above.

- **Morph between surfaces machining (HSS_MS_faces8)**

  This HSS operation uses the Morph between two adjacent surfaces strategy to define the machining of the two concave side areas on the outside of the part. This strategy generates a constant tool path that is distributed evenly on a drive surface enclosed by two check surfaces. A Ø10 mm (0.4063 in) ball nose mill is used.

- **Morph between curves machining (HSS_MC_faces11; HSS_MC_faces26; HSS_MC_faces24)**

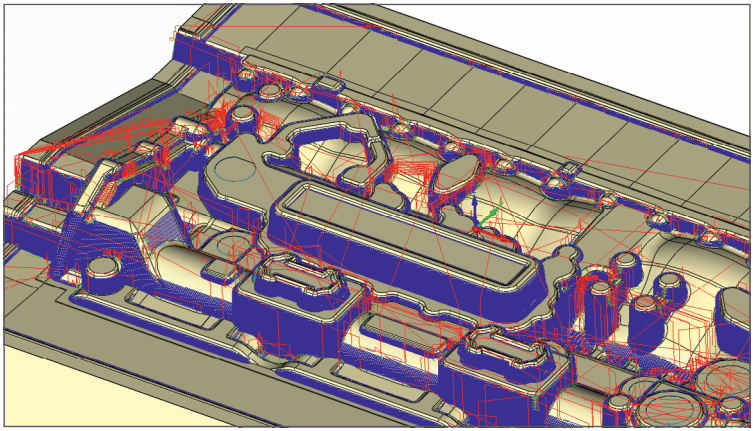
  These HSS operations use the Morph between two boundary curves strategy to define the machining of the upper and lower fillets on the external surfaces and the chamfers on the internal surfaces. This strategy generates a constant tool path that is distributed evenly between the fillet boundaries. The Ø10 mm (0.4063 in) ball nose mill is used. Gouge check is enabled to automatically detect and avoid possible collisions between the tool and the faces of the machining area.

- **Parallel to curves machining (HSS_ParC_faces13)**

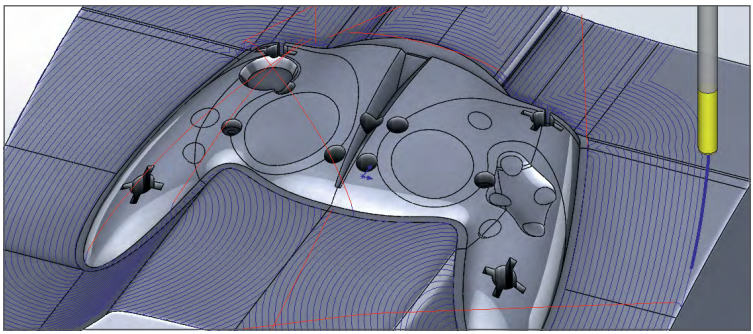
  This HSS operation uses the Parallel to curves strategy to define the machining of the lower faces on the outside of the part. With this strategy, faces can be machined with cutting passes parallel to the selected curve. In this case, a tool path is generated to follow the surfaces and to close the large U-shaped gaps by Direct linking. The Ø10 mm (0.4063 in) ball nose mill is used and the tool leads into each cut with a Vertical tangential arc.

Powerful Roughing and Finest Finish Tool Paths Available for 3D Machining!

Experience 3D machining taken to an entirely new level of smoothness, efficiency and smart machining, with the finest finish tool paths available anywhere for 3D machining.



The SolidCAM HSR/HSM module is the very powerful and market-proven 3D High Speed Machining module for complex 3D parts, aerospace parts, molds, tools and dies.



HSR/HSM offers unique machining and linking strategies for generating 3D high speed tool paths. It smoothes the paths of both cutting moves and retracts, wherever possible, to maintain a continuous machine tool motion, which is an absolute essential requirement for maintaining higher feed rates and for eliminating dwelling.

HSR – High Speed Roughing

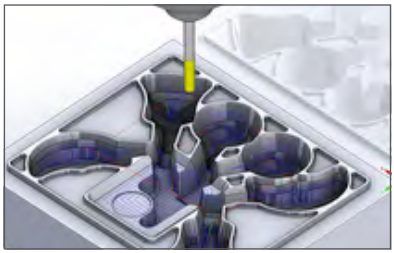
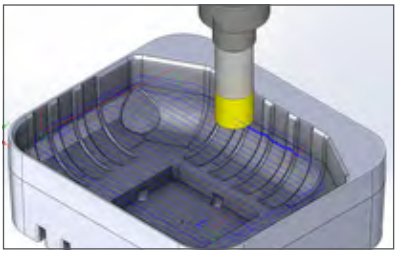
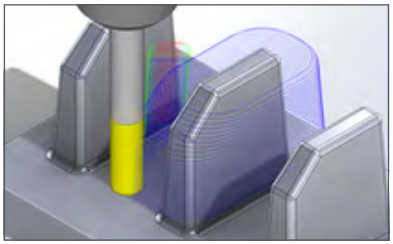
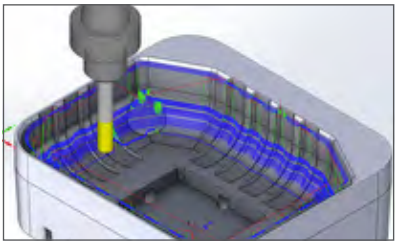
SolidCAM HSR provides powerful high speed roughing strategies such as HM, Contour, Hatch, Hybrid Rib roughing and Rest roughing.

HSM – High Speed Finishing

SolidCAM HSM keeps retracts to high Z-levels at a minimum. Angled where possible and smoothed by arcs, retracts do not go any higher than necessary, minimizing air cutting and reducing machining time.

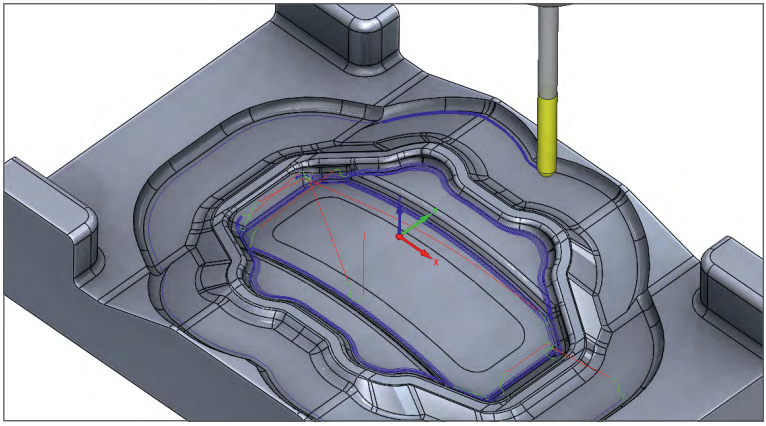
The result of HSM is an efficient and smooth tool path that translates to increased surface quality as well as less wear and longer life for your cutting tools and your machine tools.

With demands for shorter than ever lead and production times, lower costs and improved quality, SolidCAM HSM is a must in today's machine shops.

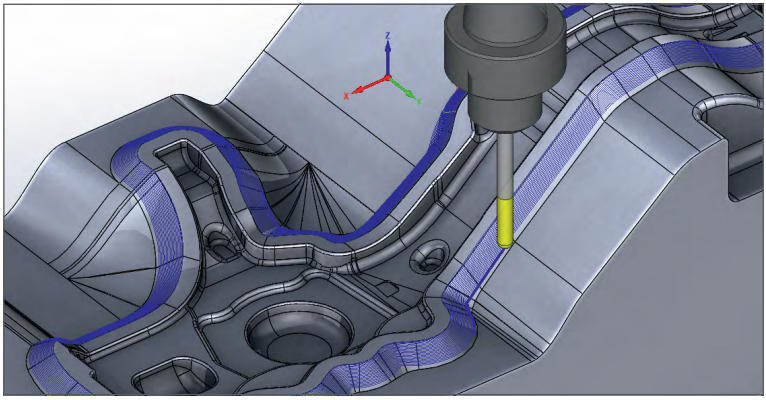





3D Mill High Speed Machining (HSR/HSM)

The SolidCAM HSR/HSM module features several enhancements to CAM technology that make high speed machining operations possible, including avoiding sharp angles in tool path, ensuring that the tool stays in contact with the material as much as possible, and optimizing non-machining moves to reduce air cutting and generating smooth and tangential lead in/out.



Any HSM strategy can be controlled by specifying the surface slope angle to be machined or by specifying the machining boundary.



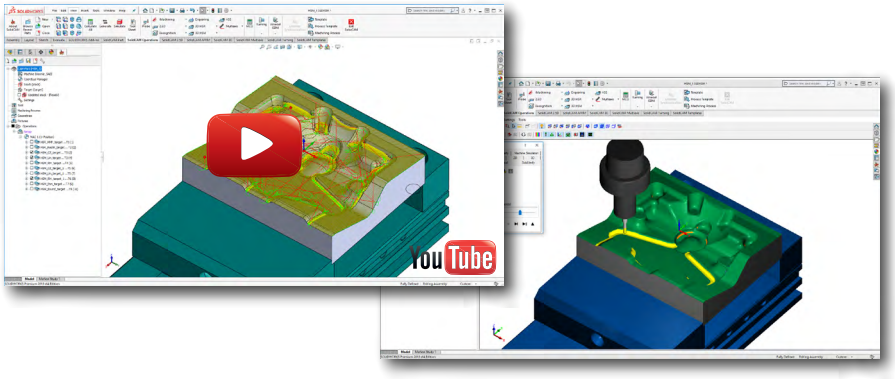
A comprehensive set of boundary creation tools is provided, including silhouette boundaries, cutter contact area boundaries, shallow boundaries, rest area boundaries and even user-defined boundaries.

HSM – 3D Machining to the Highest Level

SolidCAM HSM is the powerful solution for all users who demand advanced high speed machining capabilities. It can also be used to improve the productivity of older CNC-Machines with reduced air cutting and smoothing arcs that maintain continuous tool motion.

SolidCAM HSM can take your 3D machining performance to the highest level possible, all with your current machines.

3D Mill High Speed Machining (HSR/HSM)



Click to open the SolidCAM part: **HSM_1.prz**

Click to download the recording.

The operations in this example illustrate the use of several SolidCAM 3D Mill High Speed Machining (HSR/HSM) strategies to machine the part shown above.

- **Rough machining (HSR_HMP_target)**

  This HM Roughing operation defines the contour roughing of the mold cavity. A Ø16 mm (0.625 in) bull nose mill with a corner radius of 0.8 mm (0.0325 in) is used along with a Step down of 3 mm (0.12 in). To machine the outside and inside areas, the Constant option is chosen for the Step down type and the HM spiral option is chosen for the Step over type. A machining allowance of 0.5 mm (0.02 in) remains on the surfaces for further HSM semi-finishing and finishing operations.

- **Rest roughing (HSR_RestR_target)**

  This HSR Rest roughing operation defines the removal of unmachined material remaining after the previous operation and uses an Ø8 mm (0.3125 in) bull nose mill with a corner radius of 2 mm (0.08 in). A 1 mm (0.04 in) Step down is specified in addition to the same machining allowance as in the previous roughing operation.

- **Steep faces semi-finishing (HSM_CZ_target)**

  This HSM Constant Z machining operation defines the semi-finishing of the steep faces (from 40° to 90°). A Ø10 mm (0.4063 in) ball nose mill is used. The Apply fillets option adds virtual fillets to smooth the tool path at the corners and the machining allowance of 0.25 mm (0.01 in) remains on the surfaces for further HSM finishing operations.

- **Shallow faces semi-finishing (HSM_Lin_target)**

  This HSM Linear machining operation defines the semi-finishing of the shallow faces (from 0° to 42°). The Ø10 mm (0.4063 in) ball nose mill is used. Again, the Apply fillets option adds virtual fillets to smooth the tool path at the corners and a machining allowance of 0.25 mm (0.01 in) remains on the surfaces for further HSM finishing operations.

- **Corners semi-finishing (HSM_RM_target)**

  This HSM Rest machining operation defines the semi-finishing of the inside corners. Adding such an operation helps avoid tool overload in the corner areas before finishing. A Ø6 mm (0.2344 in) ball nose mill is used with a Reference tool of Ø16 mm (0.6299 in) to determine the rest material that requires machining. The machining allowance of 0.25 mm (0.01 in) remains on the surfaces for further HSM finishing operations.

- **Steep faces finishing (HSM_CZ_target_1)**

  This HSM Constant Z machining operation defines the finishing of the steep faces (from 40° to 90°). An Ø8 mm (0.3125 in) ball nose mill is used with the Apply fillets option.

- **Shallow faces finishing (HSM_Lin_target_1)**

  This HSM Linear machining operation defines the finishing of the shallow faces (from 0° to 42°). The Ø8 mm (0.3125 in) ball nose mill is used with the Apply fillets option.

- **Corners finishing (HSM_RM_target_1)**

  This HSM Rest machining operation defines the finishing of the inside corner areas. A Ø4 mm (0.1563 in) ball nose mill is used with a Reference tool of Ø12 mm (0.4724 in) to determine the rest material that requires machining.

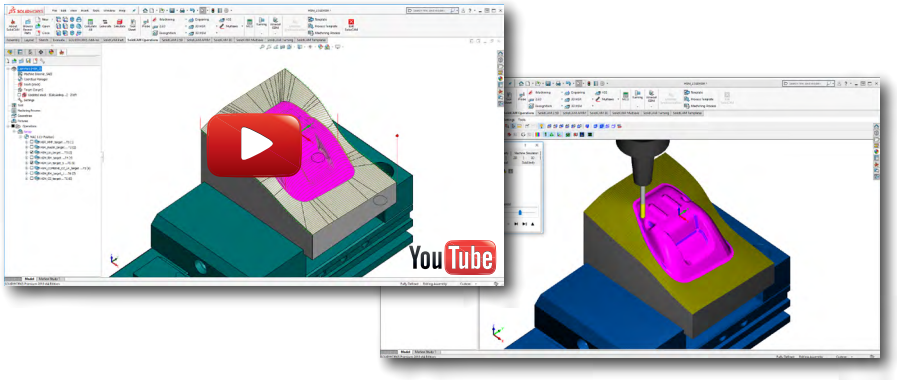
- **Corners final finishing (HSM_Pen_target)**

  This HSM Pencil milling operation defines the final finishing of the inside corner areas. A Ø6 mm (0.2344 in) lollipop mill is used to remove any material that could not be reached in the previous operations.

- **Chamfer machining (HSM_Bound_target)**

  This HSM Boundary machining operation defines the chamfering of the upper edges using a Ø10 mm (0.4083 in) chamfer drill.

3D Mill High Speed Machining (HSR/HSM)



Click to open the SolidCAM part: **HSM_2.prz**

Click to download the recording.

The operations in this example illustrate the use of several SolidCAM 3D Mill High Speed Machining (HSR/HSM) strategies to machine the part shown above.

- **Rough machining (HSR_HMP_target)**

  This HM Roughing operation defines the contour roughing of the mold core. A Ø20 mm (0.75 in) end mill is used along with a Step down of 5 mm (0.2 in). To machine the outside and inside areas, the Constant option is chosen for the Step down type and the HM spiral option is chosen for the Step over type. A machining allowance of 0.3 mm (0.012 in) remains on the surfaces for further HSM semi-finishing and finishing operations.

- **Rest roughing (HSR_RestR_target)**

  This HSR Rest roughing operation defines the removal of unmachined material remaining after the previous operation and uses a Ø12 mm (0.4844 in) ball nose mill. A 2 mm (0.08 in) Step down is specified in addition to the same machining allowance as in the previous roughing operation.

- **Semi-finishing (HSM_Lin_target)**

  This HSM Linear machining operation defines the part semi-finishing. A Ø10 mm (0.4063 in) ball nose mill is used and a 2 mm (0.08 in) Step over is specified. The Apply fillets option adds virtual fillets to smooth the tool path at the corners and a machining allowance of 0.1 mm (0.004 in) remains on the surfaces for further HSM finishing operations.

- **Corners semi-finishing (HSM_RM_target)**

  This HSM Rest machining operation defines the semi-finishing of the inside corners. Adding such an operation helps avoid tool overload in the corner areas before finishing. A Ø6 mm (0.2344 in) ball nose mill is used with a Reference tool of Ø12 mm (0.4844 in) to determine the rest material that requires machining. The machining allowance of 0.1 mm (0.004 in) remains on the surfaces for further HSM finishing operations.

- **Linear Finishing (HSM_Lin_target_1)**

  This HSM Linear machining operation defines the part finishing. A Ø6 mm (0.2344 in) ball nose mill is used with the Apply fillets option. The Constraint boundaries are created manually to limit the tool path to the core area only.

- **Combined finishing of steep and shallow areas (HSM_COMBINE_CZ_Lin_target)**

  This HSM Combine machining operation defines the Constant Z finishing of the steep areas (from 35° to 90°) and Linear finishing of the shallow areas (from 0° to 38°) according to the Constraint boundaries manually created in the previous operation. The Ø6 mm (0.2344 in) ball nose mill is used with the Apply fillets option and the Linear passes are defined by a 90° angle.

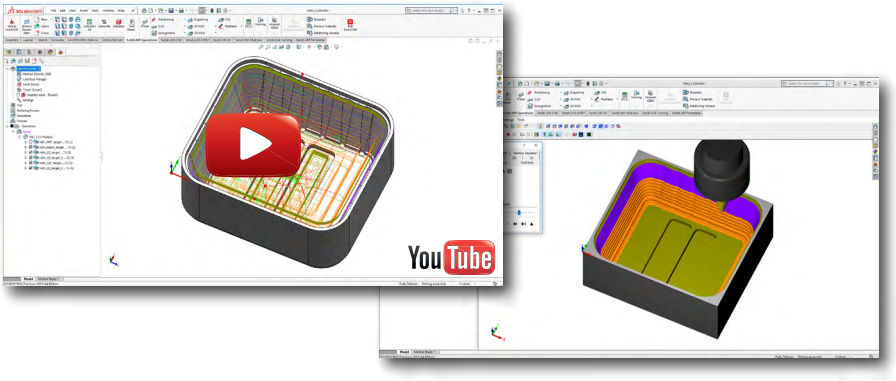
- **Corners finishing (HSM_RM_target_1)**

  This HSM Rest machining operation defines the finishing of the inside corner areas. A Ø4 mm (0.1563 in) ball nose mill is used with a Reference tool of Ø8.2 mm (0.3228 in) to determine the rest material that requires machining.

- **Concave areas machining (HSM_CS_target)**

  This HSM 3D Constant step over operation defines the finishing of the two concave areas down to a certain depth. The Ø4 mm (0.1563 in) ball nose mill is used with a step over of 0.05 mm (0.002 in) specified for both the horizontal and vertical directions.

3D Mill High Speed Machining (HSR/HSM)



Click to open the SolidCAM part: **HSM_3.prz**

Click to download the recording.

The operations in this example illustrate the use of several SolidCAM 3D Mill High Speed Machining (HSR/HSM) strategies to machine the part shown above.

- **Rough machining (HSR_HMP_target)**

  This HM Roughing operation defines the contour roughing of the electronic box. A Ø30 mm (1.1563 in) end mill is used along with a Step down of 10 mm (0.4 in). To machine the inside areas, the Constant option is chosen for the Step down type and the Cavity option is chosen for the Step over type. A machining allowance of 0.5 mm (0.02 in) remains on the surfaces for further HSM semi-finishing and finishing operations.

- **Rest roughing (HSM_RestR_target)**

  This HSR Rest roughing operation defines the removal of unmachined material remaining after the previous operation. A Ø16 mm (0.625 in) bull nose mill with a corner radius of 1 mm (0.04 in) is used. A 5 mm (0.2 in) Step down is specified in addition to the same machining allowance as in the previous roughing operation.

- **Upper faces machining (HSM_CZ_target)**

  This HSM Constant Z machining operation defines the vertical faces finishing on the upper ledge down to a certain depth. A Ø12 mm (0.4844 in) bull nose mill with a corner radius of 0.5 mm (0.02 in) is used.

- **Lower faces machining (HSM_CZ_target_1)**

  This HSM Constant Z machining operation defines the vertical faces finishing in the two lower pockets down to a certain depth. The Ø12 mm (0.4844 in) bull nose mill is used.

- **Horizontal faces machining (HSM_CZF_target)**

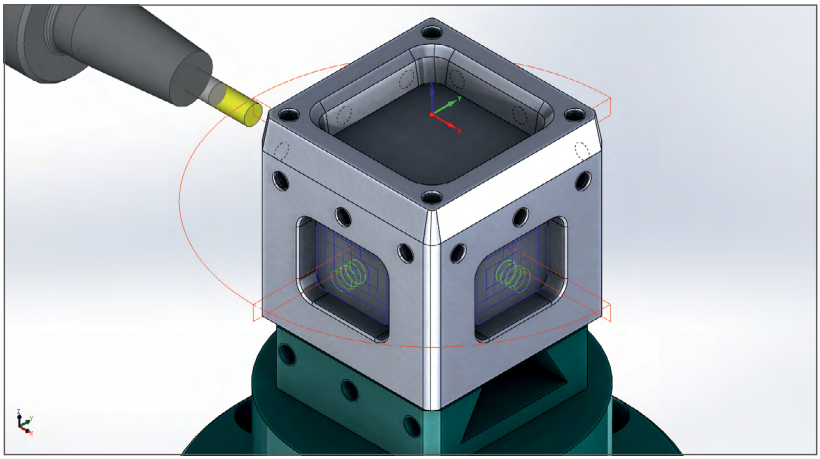
  This HSM Horizontal machining operation defines the finishing of the part horizontal faces. The Ø12 mm (0.4844 in) bull nose mill is used.

- **Inclined faces machining (HSM_CZ_target_2)**

  This HSM Constant Z machining operation defines the finishing of the part inclined faces. A taper mill is used along with a Step down of 10 mm (0.4 in). Such a tool can be used for the machining of internal or external walls with a constant draft angle.

Powerful Indexing with Multi-sided Machining – Easiest Coordinate System Definition!

The common scene in many machine shops today is that 4- and 5-Axis CNC-Machines are increasing production, providing faster cycle times.

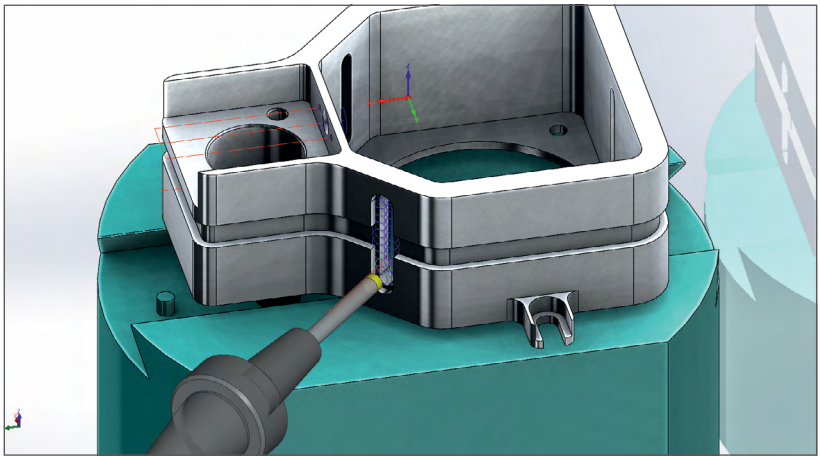


SolidCAM provides an effective and easy way to program on multiple sides of a part. SolidCAM is exceptionally strong in Indexial 4/5-Axis Machining.

Easiest Coordinate System Definition!

Tired of dealing with construction views, copying models, and rotating them in space for new alignments? Do you still copy and transform geometry to separate layers for indexial programming?

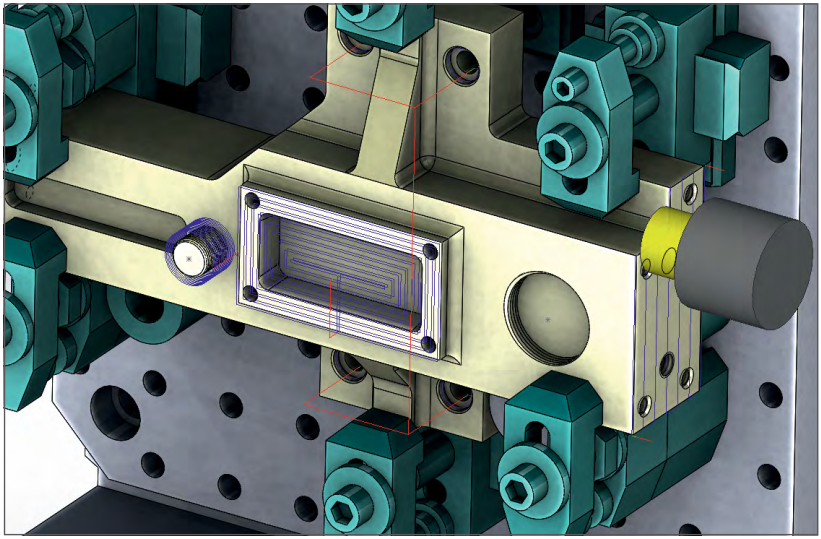
Experience single machine home positions with one-click orientations for indexed setups. SolidCAM speeds up Multi-sided Machining by eliminating multiple coordinate system constructions. Define a SolidCAM Coordinate System and/or positions on the fly, by just picking a face, and then continue programming your part.



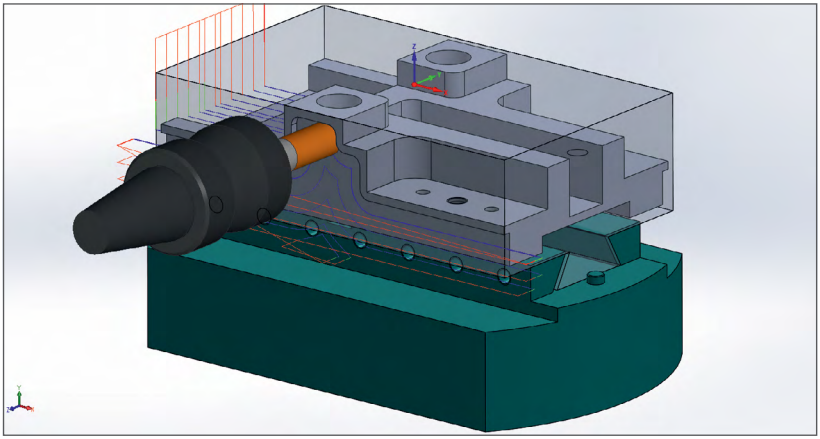
- SolidCAM's "select a face and machine" functionality is the fastest approach to indexial programming
- SolidCAM's CoordSys Manager keeps track of all necessary data for each tool orientation
- SolidCAM's SolidVerify simulation shows tool holders and fixtures, together with material removal for all machining operations

Efficient, Edit-free GCode for Multiaxis CNC-Machines

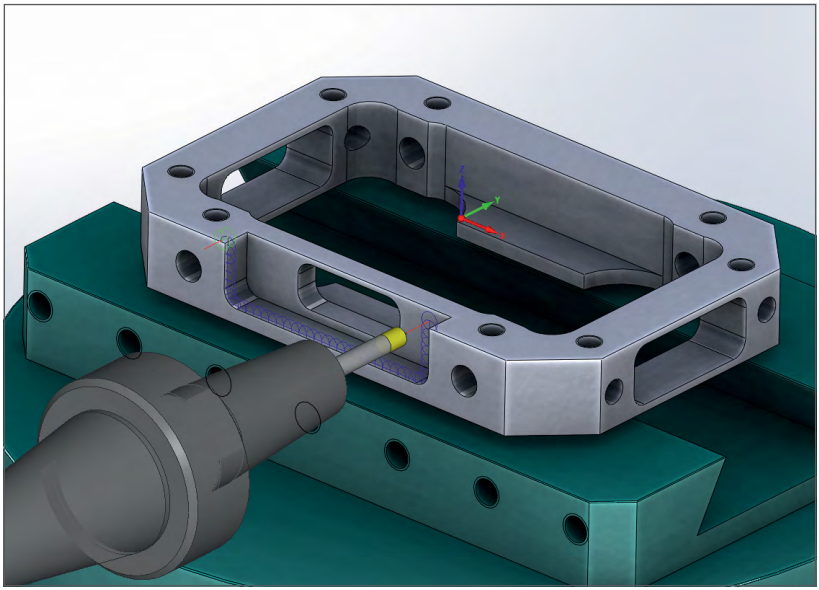
SolidCAM offers multiple options to get efficient, edit-free GCode for Multiaxis CNC-Machines.



SolidCAM's post processors can be configured to handle all rotations and work offset shifting, which eliminates the need for setting up multiple work offsets at the machine. Whether your controller can calculate part rotations internally or it requires the post processor to handle rotations, SolidCAM has it covered.

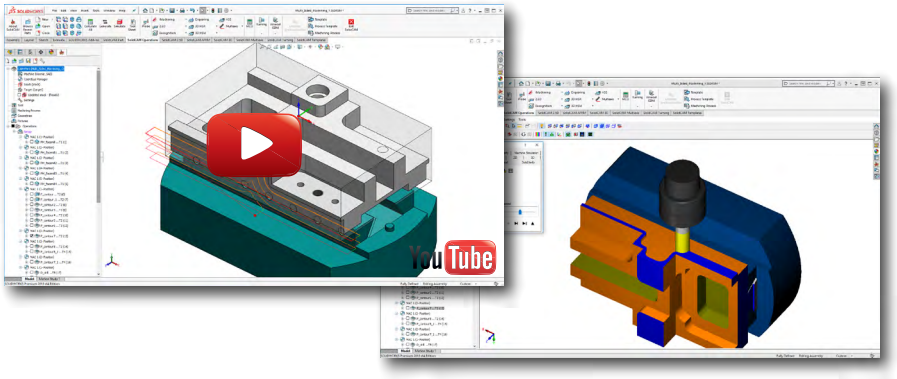


For controllers with advanced plane rotation or coordinate rotation functions, SolidCAM's post processors are built to use these internal CNC functions. If you have a machine without such functions, users can input the part location inside SolidCAM and the GCode will handle all the GCode transformations for each rotation.



SolidCAM's philosophy towards Indexial 4/5-Axis Machining is simple: from software to GCode, make the process for Multi-sided Machining the same as for Single-sided Machining. There is no need for any special functions or tricks inside the software to machine multi-sided parts – it should just work!

Indexial Multiaxis Machining



Click to open the SolidCAM part: **Multi_Sided_Machining_1.prz**

Click to download the recording.

The operations in this example illustrate the use of SolidCAM's Indexial Multiaxis Machining capabilities to machine the part shown above. The machining is performed on a 5-Axis CNC-Machine in one setup, using one SolidCAM Coordinate System with multiple positions.

- **Top face machining (FM_facemill)**

  This Face Milling operation defines the machining of the part top face using Machine Coordinate System #1, Position #1 (MAC 1 (1- Position)).

- **Side faces machining (FM_facemill1; FM_facemill2; FM_facemill3; FM_facemill4)**

  These Face Milling operations define the machining of the part side faces using MAC 1 (2, 3, 4 and 5- Positions).

- **Step machining (F_contour; F_contour_1)**

  These Profile operations define the rough and finish machining of the step feature using MAC 1 (1- Position).

- **Open pocket machining (P_contour2)**

  This Pocket operation defines the machining of the open pocket located below the step feature using MAC 1 (1- Position).

- **Through pocket machining (P_contour3)**

  This Pocket operation defines the machining of the through pocket located below the step feature using MAC 1 (1- Position).

- **Open pockets machining (P_contour4; P_contour5; P_contour6)**

  These Pocket operations define the machining of the three remaining open pockets using MAC 1 (1- Position).

- **Side pockets machining (P_contour7; P_contour8)**

  These Pocket operations define the machining of the open pockets located on the opposing side faces using MAC 1 (2 and 3- Positions).

- **Corners rest and finish machining (P_contour8_1; P_contour7_1)**

  These Pocket operations define the inside corners machining of the open pockets created in the previous operations using the same MAC 1 (2 and 3- Positions).

- **Holes machining (D_drill – D_drill2_2)**

  These Drilling operations define the center drilling, the drilling and tapping of the holes located on all the part faces using MAC 1 (1- Position).

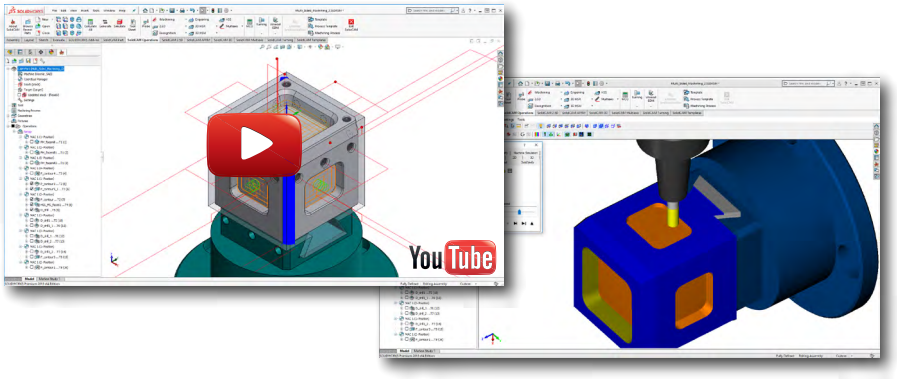
- **Counterbore machining (F_contour15_1)**

  This Profile operation defines the machining of the two counterbore features using MAC 1 (1- Position).

- **Chamfer machining (F_contour9 – F_contour18)**

  These Profile operations define the machining of the chamfers on the part edges using MAC 1 (1, 2, 3, 4 and 5- Positions).

Indexial Multiaxis Machining



Click to open the SolidCAM part: **Multi_Sided_Machining_2.prz**

Click to download the recording.

The operations in this example illustrate the use of SolidCAM's Indexial Multiaxis Machining capabilities to machine the part shown above. The machining is performed on a 5-Axis CNC-Machine in one setup, using one SolidCAM Coordinate System with multiple positions.

- **Top face machining (FM_facemill)**

  This Face Milling operation defines the machining of the part top face using Machine Coordinate System #1, Position #1 (MAC 1 (1- Position)).

- **Side faces machining (FM_facemill1)**

  This Face Milling operation defines the machining of the part side faces using MAC 1 (2- Position). The Transform 4th Axis option is used to create a circular pattern of operations around the revolution axis.

- **Inclined faces machining (FM_facemill3)**

  This Face Milling operation defines the machining of the inclined faces using MAC 1 (5- Position). The Transform 4th Axis option is used again to create a circular pattern of operations around the revolution axis.

- **Corner faces machining (F_contour4)**

  This Profile operation is defined to machine the vertical and inclined faces of two opposing corners using MAC 1 (4- Position). The Transform 4th Axis option is used again to create a circular pattern of operations around the revolution axis.

- **Top pocket machining (P_contour3; F_contour5_1)**

  These Pocket and Profile operations define the machining of the top pocket and its inclined faces using MAC 1 (1- Position).

- **Side features machining (P_contour; HSS_MS_faces1; D_drill)**

  These operations define the machining of several side features using MAC 1 (2- Position). For each operation, the Transform 4th Axis option is used to create a circular pattern of relevant operations around the revolution axis. The Pocket operation machines the pockets. The HSS Morph between two adjacent surfaces operation machines the external fillets on the two opposing corners not yet machined. The Drilling operation defines the center drilling of the three holes located on each of the side faces.

- **Top holes machining (D_drill1; D_drill1_1; D_drill1_2)**

  These Drilling operations define the center drilling, the drilling and tapping of the holes located on the part top face using MAC 1 (1- Position).

- **Side holes machining (D_drill_1; D_drill_2)**

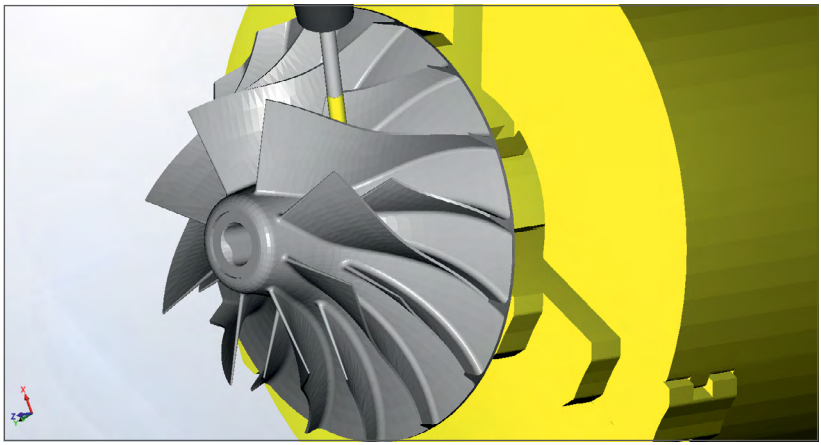
  These Drilling operations define the drilling and tapping of the holes located on the part side faces using MAC 1 (2- Position). The Transform 4th Axis option is used again to create a circular pattern of operations around the revolution axis.

- **Chamfer machining (F_contour5; F_contour1)**

  These Profile operations define the machining of the chamfers on the top and side pocket edges using MAC 1 (1 and 2- Positions). For the last operation, the Transform 4th Axis option is used again to create a circular pattern of operations around the revolution axis.

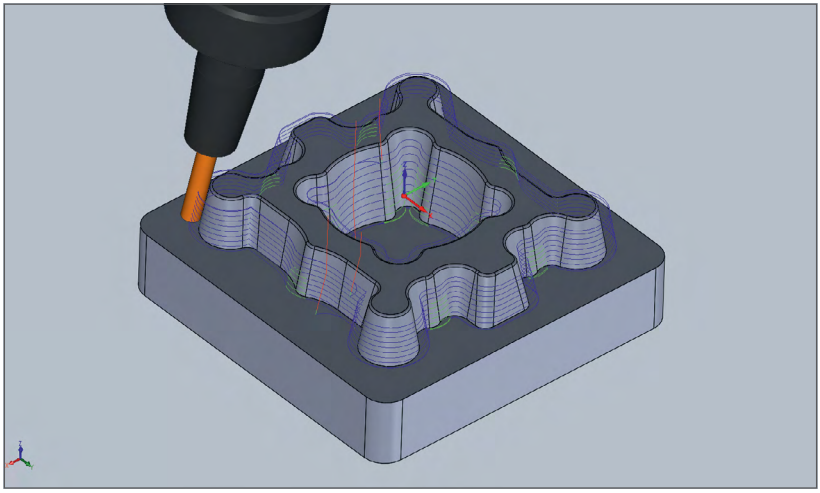
Powerful Simultaneous 5-Axis Tool Paths with a Very Friendly User Interface

Benefit from the most tested and most proven 5-Axis machining tool paths in the industry with a user-friendly interface, collision checking and the most advanced control over all aspects of tool path.



- Wide variety of Simultaneous 5-Axis Machining strategies
- Flowline cutting produces a tool path that follows the natural shape of the component
- Multi-surface finish machining keeps the tool normal to the surface (or with specified lead and lag) to provide a smooth surface finish
- Specific applications solutions for SWARF, Multiblade, Port, Contour 5-Axis, Multiaxis Drilling and conversion of HSM to Sim. 5-Axis
- Advanced tool tilting control and direct control on side tilting and lead/lag angles
- Automatic collision avoidance strategies that check each part of both the tool and holder
- Realistic full 3D Machine Simulation with comprehensive collision and axes limits checking

SolidCAM's Simultaneous 5-Axis Machining supports all Multiaxis machine tools including Head/Head, Table/Table and Head/Table Gantry Machines as well as the latest Mill-Turn Machining Centers.



Flexibility and Control

Each Simultaneous 5-Axis Machining strategy provides sophisticated options for approach/link control and tool axis control.

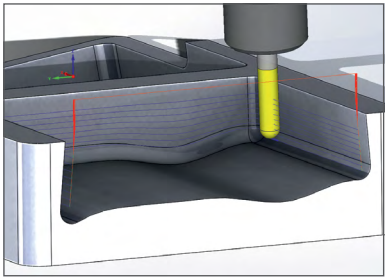
Link and approach movements are fully protected from gouging and different strategies may be used depending on the distance of the link move. SolidCAM also provides options for control over lead/lag and side tilt angles to give complete control over the final tool path.

Collision Avoidance for Tool and Holder

Collision avoidance is supported for both the tool and holder, and a range of strategies is offered for avoiding collisions. The Machine Simulation provides complete visualization of the gouge checking.

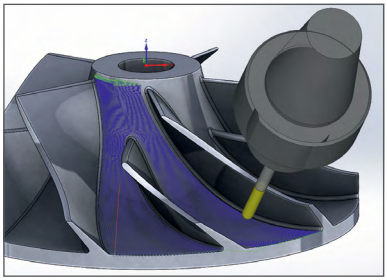
SWARF Machining

SWARF Machining allows the side of the tool to be tilted over to machine the side wall at the correct angle. The SWARF operation uses the whole cutting length of the tool, resulting in better surface quality and shorter machining time.



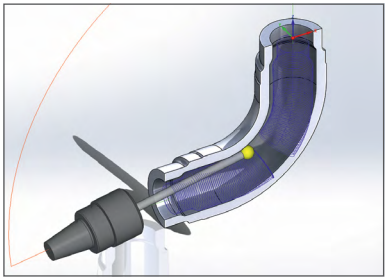
Multiblade Machining

Multiblade Machining easily handles impellers and bladed disks with several strategies to efficiently rough and finish all parts of these complex shapes. Multiple bladed parts are used in many industries and this operation is specifically designed to generate the necessary tool path for the different configurations.



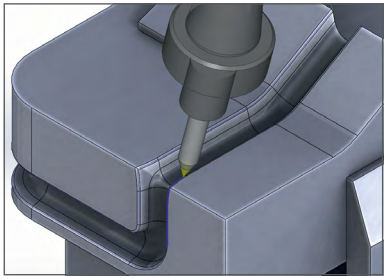
Port Machining

Port Machining is an easy-to-use method for machining ports with tapered lollipop tools, which can reach the full area while offering collision checks for the entire tool. Several strategies are provided to generate roughing and finishing tool paths for making ports from castings or billets.



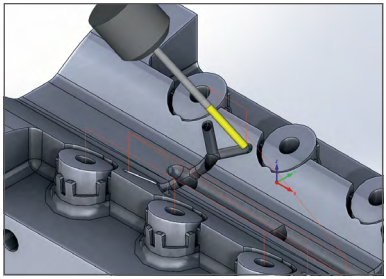
Contour 5-Axis Machining

Contour 5-Axis Machining tilts the tool along a chained 3D profile drive curve, while aligning the tool axis according to defined tilt lines, making it ideal for generating 5-Axis tool path for deburring and trimming.



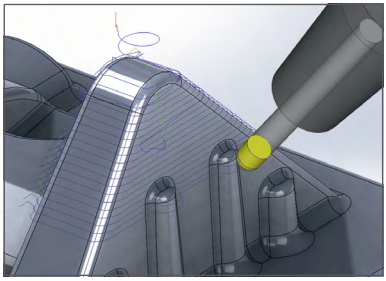
Multiaxis Drilling

Multiaxis Drilling uses SolidCAM's Drill Recognition to automatically recognize all the part hole features and then perform drilling, tapping or boring cycles at any hole direction easily and quickly. In this operation, all the advanced linking, tilting and collision checking are available.

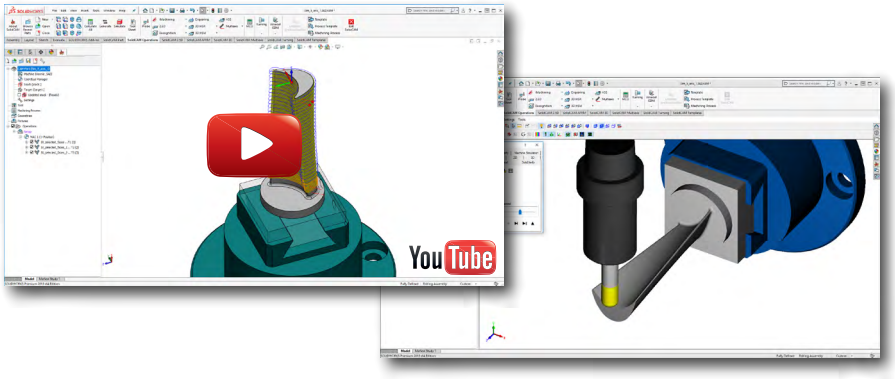


HSM to Sim. 5-Axis Milling

3 to 5 axis Conversion converts HSM 3D tool paths to full 5-Axis collision-protected tool paths. This will maintain optimum contact point between the tool and the part and enable the use of shorter tools for more stability and rigidity.



Simultaneous 5-Axis Machining



Click to open the SolidCAM part: **Sim_5_axis_1.prz**

Click to download the recording.

The operations in this example illustrate the use of SolidCAM's Simultaneous 5-Axis Machining to semi-finish and finish the turbine blade part shown above.

- **Blade Semi-finishing (5X_selected_faces; 5X_selected_faces_1)**

  These Sim. 5-Axis Milling operations use the Parallel cuts strategy to define the semi-finishing of the part.

  The first operation provides the semi-finish for the turbine blade surfaces near the end using a Ø16 mm (0.625 in) bull nose mill with a corner radius of 4 mm (0.16 in). Tilted relative to cutting direction defines the tool tilting with a lag angle of 20° and the front face of the tool is specified for the tool contact point. This combination of parameters performs the machining trochoidally by the end surface of the tool. Gouge checking is enabled to detect and avoid possible collisions between the tool, the geometry drive surfaces and the check surfaces of the turbine blade base.

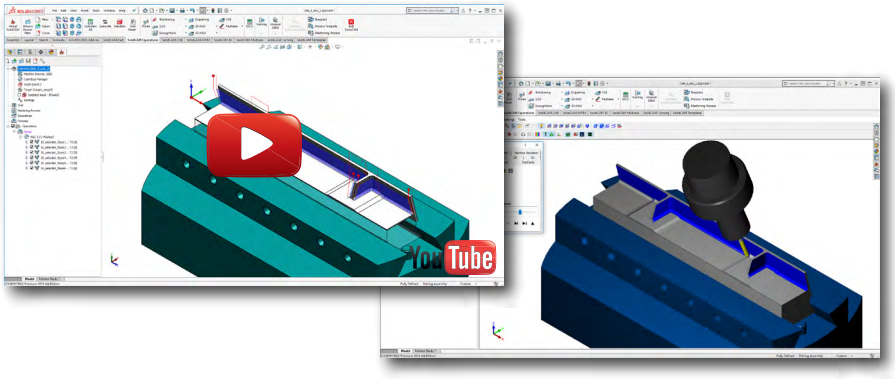
  The second operation provides the semi-finish for the turbine blade surfaces near the base. This area was not machined in the previous operation because of the gouge protection. An Ø8 mm (0.3125 in) bull nose mill with a corner radius of 2 mm (0.08 in) is used. Tilted relative to cutting direction defines the tool tilting with a lag angle of 20° and a side tilting angle of 10°. The front face of the tool is specified for the tool contact point. The same options are enabled for gouge checking as in the previous operation.

  The remaining material from both operations will be removed during the finish machining.

- **Blade finishing (5X_selected_faces_2)**

  This Sim. 5-Axis Milling operation uses the Parallel cuts strategy to define the finishing of the turbine blade surfaces. An Ø8 mm (0.3125 in) bull nose mill with a corner radius of 2.5 mm (0.1 in) is used. Tilted relative to cutting direction defines the tool tilting with a lag angle of 20° and a side tilting angle of 10°. The front face of the tool is specified for the tool contact point. Gouge checking is enabled to detect and avoid possible collisions between the tool and the check surfaces of the turbine blade base. The tool path is generated by a number of cuts that are parallel between the end and base of the turbine blade.

Simultaneous 5-Axis Machining



Click to open the SolidCAM part: **Sim_5_axis_2.prz**

Click to download the recording.

The operations in this example illustrate the use of SolidCAM's Simultaneous 5-Axis Machining to finish the aerospace part shown above.

- **Inclined walls finishing (5X_selected_faces1; 5X_selected_faces2; 5X_selected_faces3)**

  These Sim. 5-Axis Milling operations use the Parallel cuts strategy to define the finishing of the inclined walls, which are forming undercut areas.

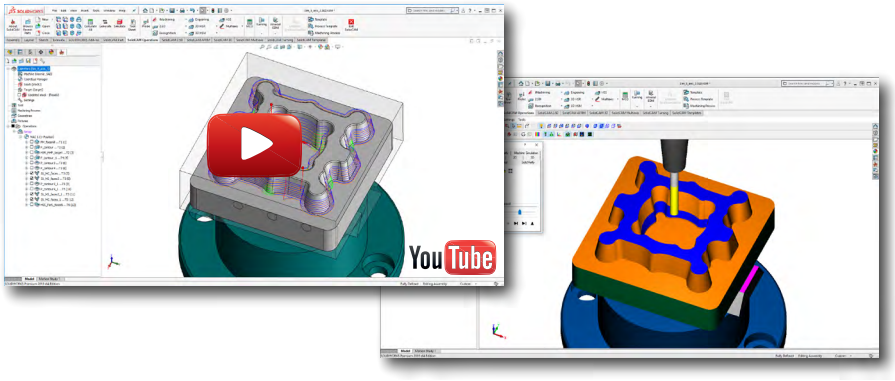
  The three operations provide the finish for the inclined walls that partially surround each area using a Ø4 mm (0.1563 in) ball nose mill. Aside from the geometry selection differences, the Operation definitions are the same. Tilted relative to cutting direction defines the tool tilting with a side tilting angle of 90°. Auto is specified for the tool contact point. This combination of parameters performs the machining axially by the side surface of the tool. For each operation, gouge checking is enabled to detect and avoid possible collisions between the tool, the geometry drive surfaces and the check surfaces of the adjacent fillets. The tool path is generated by a number of cuts parallel to each other along the inclined walls.

- **Fillet finishing**
  (5X_selected_faces4; 5X_selected_faces5; 5X_selected_faces6)

  These Sim. 5-Axis Milling operations use the Projection (User-defined) strategy to define the finishing of the fillets adjacent to the inclined walls.

  The three operations provide the finish for the adjacent fillets in each area using the Ø4 mm (0.1563 in) ball nose mill. Aside from the geometry selection differences, the Operation definitions are the same. Tilted through curve defines the tool tilting with a variable tilting angle. Auto is specified for the tool contact point. This combination of parameters performs a smooth transition between different tool axis orientations. For each operation, gouge checking is enabled to detect and avoid possible collisions between the tool and the geometry drive surfaces. The tool path is generated by a single pass that follows the lower curves of the fillets projected on the drive surfaces.

Simultaneous 5-Axis Machining



Click to open the SolidCAM part: **Sim_5_axis_3.prz**

Click to download the recording.

The operations in this example illustrate the use of SolidCAM's Simultaneous 5-Axis Machining to semi-finish and finish both the internal and external inclined walls of the machine part shown above.

- **Rough machining (FM_facemill; F_contour; HSR_HMP_target; P_contour3; P_contour4)**

  These operations define the rough machining of the entire part (i.e., roughing of the upper face and outside shape, internal and external inclined walls as well as the open and closed pockets).

- **Internal inclined walls semi-finishing (5X_MC_faces)**

  This Sim. 5-Axis Milling operation uses the Morph between two boundary curves strategy to define the semi-finishing of the internal inclined walls. An Ø8 mm (0.3125 in) end mill is used and a Drive surface offset of 0.2 mm (0.008 in) is specified. Tilted relative to cutting direction defines the tool tilting with a side tilting angle of 90°. Auto is specified for the tool contact point. This combination of parameters performs the machining axially by the side surface of the tool. Gouge checking is enabled to detect and avoid possible collisions between the tool, the geometry drive surfaces and the check surfaces of the pocket floor. A morphed tool path is generated along the geometry drive surfaces and between the selected start edge and end edge curves.

- **External inclined walls semi-finishing (5X_MS_faces2)**

  This Sim. 5-Axis Milling operation uses the Morph between two adjacent surfaces strategy to define the semi-finishing of the external inclined walls. An Ø8 mm (0.3125 in) end mill is used and a Drive surface offset of 0.2 mm (0.008 in) is specified. Tilted relative to cutting direction defines the tool tilting with a side tilting angle of 90°. Auto is specified for the tool contact point. This combination of parameters performs the machining axially by the side surface of the tool. Gouge checking is enabled to detect and avoid possible collisions between the tool and the geometry drive surfaces. A morphed tool path is generated along the geometry drive surfaces and between the selected start edge and end edge surfaces.

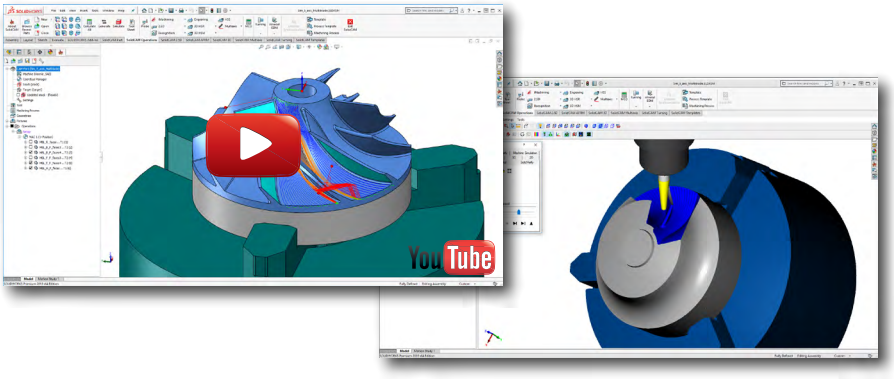
- **Finish machining (F_contour_1; P_contour3_1; P_contour4_1; HSS_ParS_faces6)**

  These operations define the finish machining of the part, except for the internal and external inclined walls (i.e., finishing of the outside shape, pocket floors and the fillets).

- **Internal and external inclined walls finishing (5X_MS_faces2_1; 5X_MC_faces_1)**

  These Sim. 5-Axis Milling operations are defined to finish the internal and external inclined walls. The two previous Sim. 5-Axis Milling operations were simply saved and copied to create these finishing operations with the same parameters. Another Ø8 mm (0.3125 in) end mill is used to perform the finish machining and a value of 0 is specified for the Drive surface offset. Again, the morphed tool path is generated along the geometry drive surfaces.

Simultaneous 5-Axis Machining



Click to open the SolidCAM part: **Sim_5_axis_Multiblade.prz**

Click to download the recording.

The operations in this example illustrate the use of SolidCAM's Multiblade Machining strategies to machine a section of the multiple bladed part shown above.

- **Roughing (MBL_R_faces)**

  This Roughing operation defines the rough machining of an area between two main blades using a Ø12 mm (0.4844 in) taper mill. For both Layers and Slices, 10 is specified as the maximum number permitted in the tool path. Tool tilting is controlled according to a preferred lead angle of 5° and a maximum lead angle of 45°. A morphed tool path is generated between the hub and shroud, gradually removing the material by layers and slices.

- **Blade finishing (MBL_B_F_faces3; MBL_B_F_faces4)**

  These Blade finishing operations define the finish machining of two blades (one main and one splitter) using a Ø12 mm (0.4844 in) taper ball nose mill. Aside from the geometry selection differences, the Operation definitions are the same. Full (trim trailing edge) is chosen so the tool follows the contour of the blades but does not roll around the trailing edges. A maximum number of 15 layers is permitted in the tool path. Tool tilting is controlled according to a preferred lead angle of 5° and a maximum lead angle of 45°. A morphed tool path is generated between the hub and shroud, gradually removing the rest material by layers along the contours of each blade.

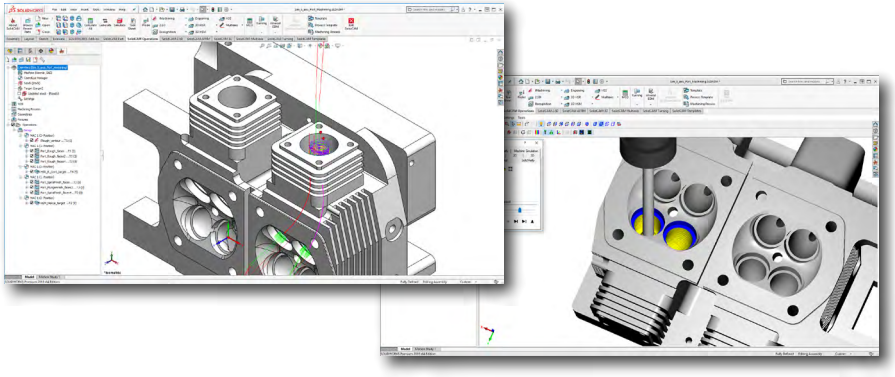
- **Fillet finishing (MBL_F_F_faces3; MBL_F_F_faces4)**

  These Fillet finishing operations define the finish machining of the fillets adjacent to the hub for the previously finished main and splitter blades. The Ø12 mm (0.4844 in) taper ball nose mill is used. Aside from the geometry selection differences, the Operation definitions are the same. Full (trim trailing edge) is chosen so the tool follows the contour of the fillets but does not roll around the trailing edges. 8 cuts are specified along the blade side and 5 cuts are specified along the hub side with a 0.3 mm (0.012 in) side step on both sides. The same angles are defined for the tool tilting as in the previous operations. SolidCAM automatically detects the fillets and the finishing tool path is generated accordingly.

- **Hub finishing (MBL_H_F_faces)**

  This Hub finishing operation defines the finish machining of the hub area surrounding the previously finished splitter blade. The Ø12 mm (0.4844 in) taper mill is used. A maximum number of 18 slices is permitted in the tool path and the same angles are defined for the tool tilting as in the previous operations. This combination of parameters enables the tool to achieve a smooth tool path around the splitter blade and along the hub surface.

Simultaneous 5-Axis Machining



Click to open the SolidCAM part: **Sim_5_axis_Port_Machining.prz**

The operations in this example illustrate the use of SolidCAM's Port Machining strategies to machine internal surfaces of intakes and curved tubes.

- **Pockets roughing (iRough_contour)**

  This iMachining operation defines the rough machining of the top part of two curved tubes using a Ø10 mm (0.4063 in) end mill. The tool performs a helical entry into each of the pockets and then, in 1 Step down, spirals to the outer walls. A machining allowance of 0.05 mm (0.002 in) remains for further finishing operations.

- **Curved tubes roughing (Port_Rough_faces; Port_Rough_faces2)**

  These Roughing operations define the rough machining of the internal surfaces of two curved tubes using a Ø16 mm (0.625 in) lollipop mill. Aside from the geometry selection differences, the Operation definitions are the same. Tool path parameters are specified with a 5 mm (0.2 in) Maximum step over and a 2 mm (0.08 in) Step down. Degouging is enabled to detect and avoid possible collisions between the tool, the adjacent outer surfaces and against any machined surfaces. The tool path is output in 5-Axis format, gradually removing the material within the tubes by layers and slices until the Maximum from top distance is reached.

- **Port roughing (Port_Rough_faces4)**

  This Roughing operation defines the rough machining of the port connecting the two curved tubes using the Ø16 mm (0.625 in) lollipop mill. The same Maximum step over and Step down parameters are specified as in the previous operations. Degouging is enabled to detect and avoid possible collisions between the tool, the adjacent outer surfaces and against any machined surfaces. The tool path is output in 5-Axis format, gradually removing the material within the port by layers and slices until the Maximum from bottom distance is reached.

- **Curved tubes finishing (Port_SpiralFinish_faces; Port_PlungeFinish_faces2)**

  These operations use the Spiral finishing and Plunge finishing strategies to define the finish machining of similar geometries. Both operations use the Ø16 mm (0.625 in) lollipop mill and aside from the geometry selection differences, the Operation definitions are the same. A Maximum step over of 2 mm (0.08 in) is specified and the same Degouging options are used as in the previous roughing operations. The tool path is generated using spiral cuts in one curved tube and plunge cuts in the other to finish the internal surfaces until the Maximum from top distance is reached.

- **Port finishing (Port_SpiralFinish_faces4)**

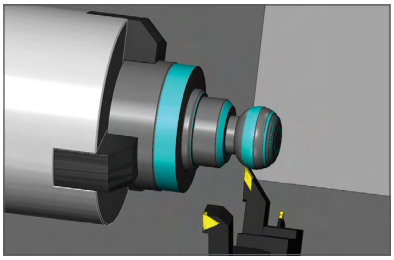
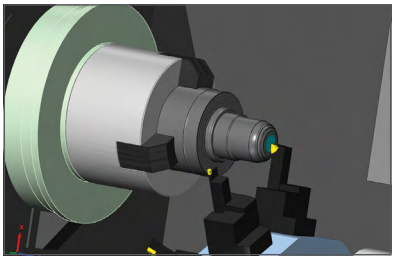
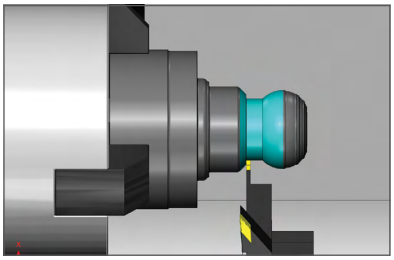
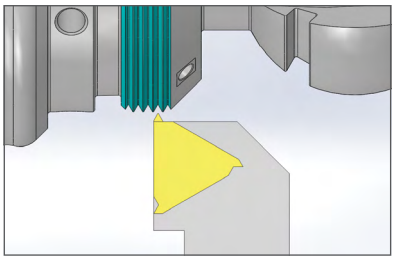
  This Spiral finishing operation defines the finish machining of the port connecting the two curved tubes using the Ø16 mm (0.625 in) lollipop mill. A 1 mm (0.04 in) Maximum step over is specified and the same Degouging options are used as in the previous roughing operation. The tool path is generated using spiral cuts in the port to finish the internal surfaces until the Maximum from bottom distance is reached.

SolidCAM Module for Fast and Efficient Turning

SolidCAM provides the most comprehensive Turning package with powerful tool paths and techniques for fast and efficient turning.

SolidCAM Turning provides functionality for a wide range of machine tools, including 2-Axis Lathes, Multi-turret Configurations, Sub-spindle Turning Centers and Mill-Turn CNC-Machines.

SolidCAM produces advanced rough and finish profile turning, together with support for facing, grooving, threading and drilling in either canned cycles or long GCode.

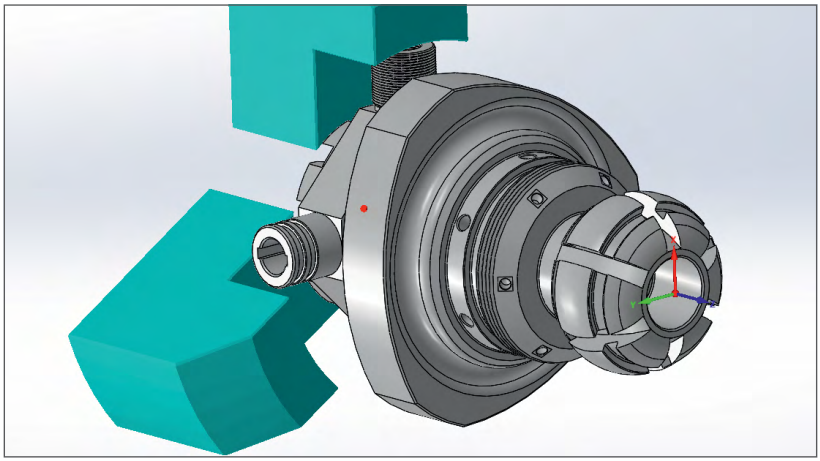





Tool path calculation takes into consideration the tooling insert, tool holder and previously machined stock material, to avoid gouging and to eliminate air cutting.

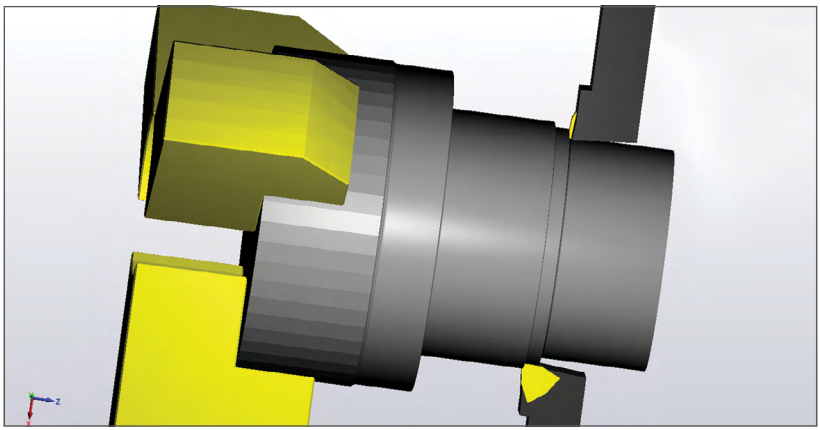
Standard fixture libraries are available and specialized fixtures can be added.

Updated Stock

SolidCAM has the ability to keep the stock updated live within the operations tree. Updated stock is supported from the most basic 2-Axis Turning Center, right through to a CYB Multi-turret, Sub-spindle Mill-Turn CNC-Machine.



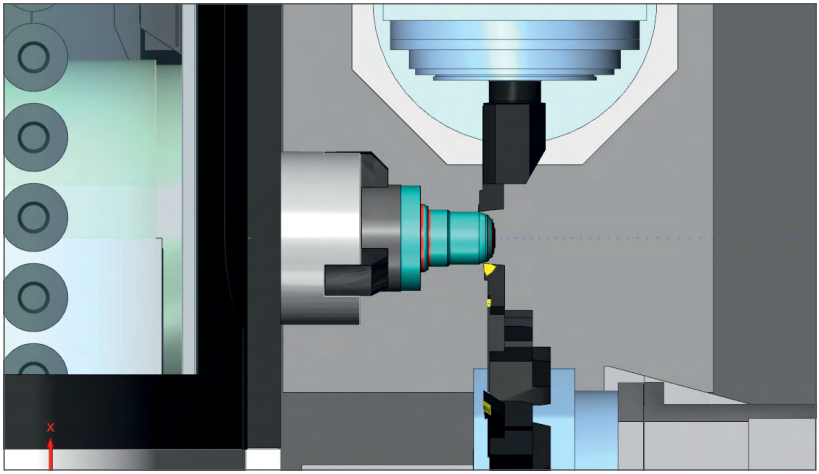
On a Sub-spindle Turning Center, when a component is transferred from the main to the sub-spindle, the Updated stock model is transferred with it. Any subsequent machining on the sub-spindle will detect the stock in the state that it left the main spindle, ultimately providing the most efficient machining sequence possible.



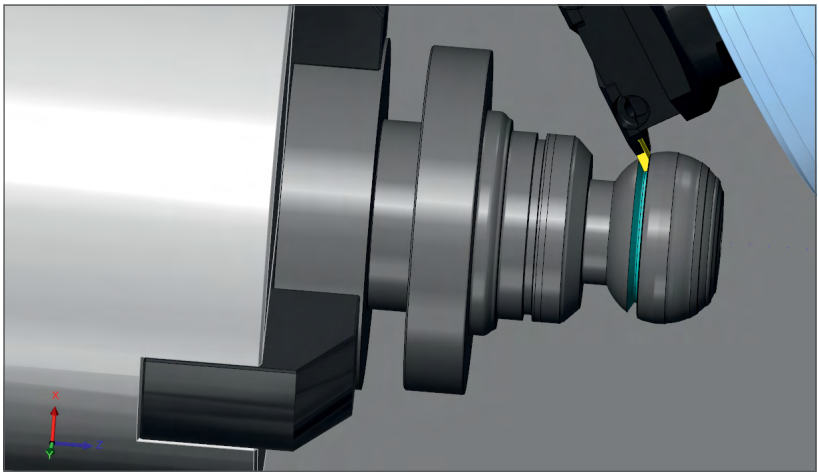
Advanced Turning Operations

SolidCAM's advanced operations bring power to all turning applications.

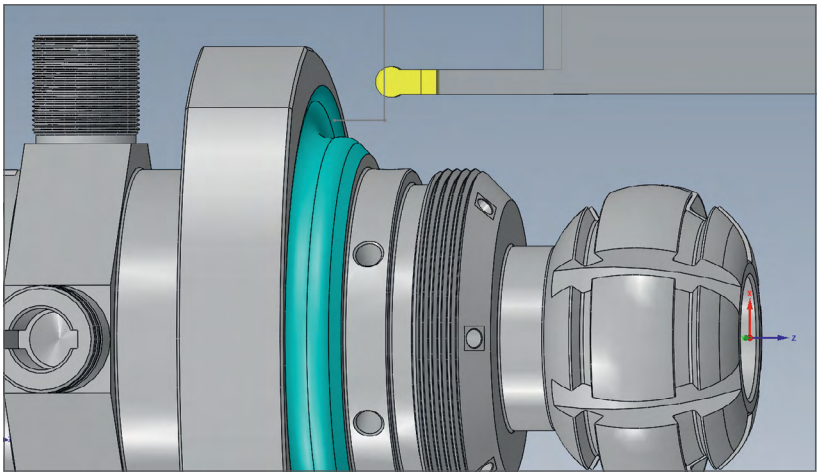
- Balanced Roughing: two tools working simultaneously or in trailing mode to perform the roughing of long and/or large parts



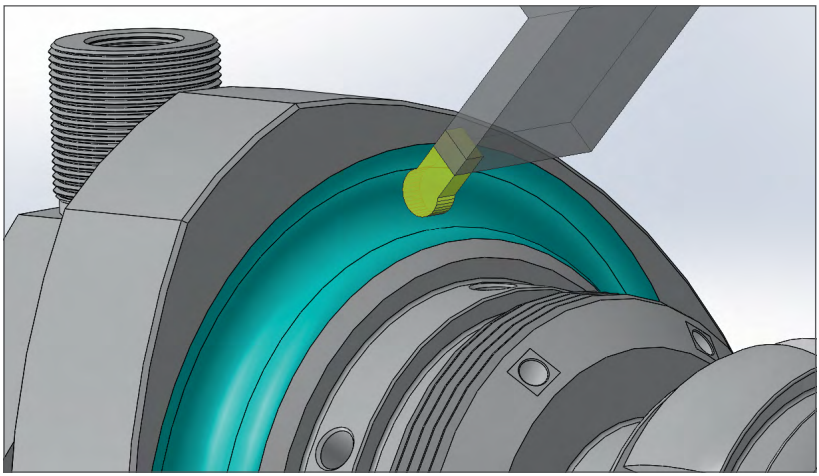
- Angled Grooving: performs internal or external inclined grooves at any defined angle



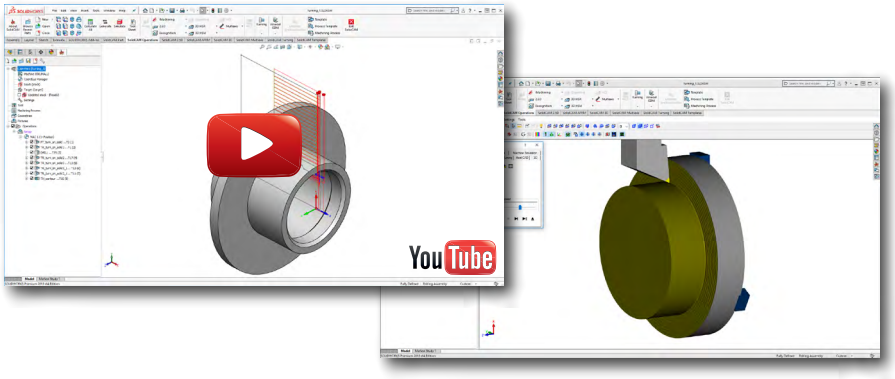
- Manual Turning: performs turning according to user-defined geometry regardless of Stock and Target models



- 4th Axis Simultaneous Turning: performs machining of curved profiles using the B-Axis tilting capabilities of the tool in order to machine undercut areas in a single machining step



- Multi-turret Synchronization: provides powerful functionality with a machining timeline to perform synchronization of operations on Multi-turret Machines



Click to open the SolidCAM part: **Turning_1.prz**

Click to download the recording.

The operations in this example illustrate the use of several SolidCAM Turning technologies to machine the part shown above.

- **Face turning (FT_turn_on_solid)**

  This Face Turning operation defines the rough machining of the front face using an external roughing tool. A 0.1 mm (0.004 in) Z-Offset remains on the surface for further finishing operations.

- **Contour turning (TR_turn_on_solid1)**

  This Turning operation defines the machining of the external surfaces on the front side using an external roughing tool. The Rough option is chosen for the work type to perform the machining in a number of equidistant passes. Long external mode is used to create the passes above the geometry and along the rotational axis of the part. The specified roughing offsets are removed in the same operation with a finishing cut.

- **Drilling (DRILL)**

  This Drilling operation defines the drilling of the hole through the center of the part using a Ø23 mm (0.9063 in) flat drill.

- **Internal roughing (TR_turn_on_solid2)**

  This Turning operation defines the rough machining of the internal surfaces including the chamfer using an internal roughing tool. The Rough option is chosen for the work type and Long internal mode is used to create passes under the geometry and along the rotational axis of the part. Non-descending motions strategy is used to avoid collisions.

- **Internal grooving (TR_turn_on_solid3)**

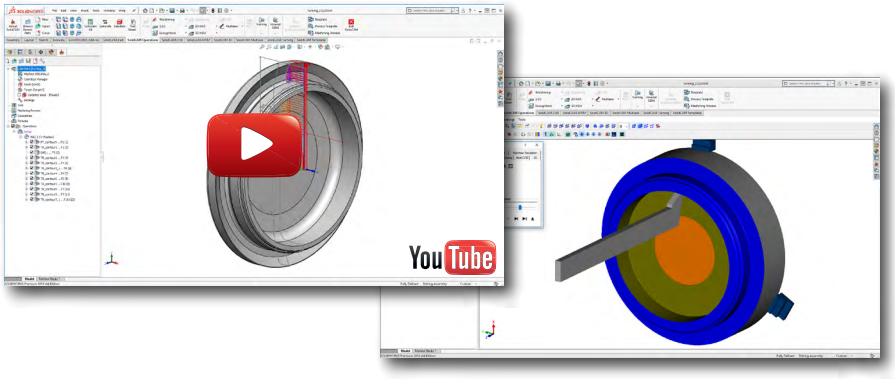
  This Turning operation defines the rough machining of the internal groove using an internal grooving tool. The Smooth option is chosen for the roughing type to provide a smoother finish.

- **Internal finishing (TR_turn_on_solid3_1; TR_turn_on_solid2_1)**

  These Turning operations define the finish machining of the internal groove and the internal surfaces using the appropriate tools. The Finish only option is chosen for the work type to perform only finishing cuts.

- **Internal threading (TH_contour)**

  This Threading operation defines the machining of an internal thread using an internal threading tool. A user-defined Thread size and Pitch unit are specified with no thread finishing.



Click to open the SolidCAM part: **Turning_2.prz**

Click to download the recording.

The operations in this example illustrate the use of several SolidCAM Turning technologies to machine the part shown above.

- **Face turning (FT_contour5)**

  This Face Turning operation defines the rough machining of the front face using an external roughing tool. A 0.2 mm (0.008 in) Z-Offset remains on the surface for further finishing operations.

- **Contour roughing (TR_contour1)**

  This Turning operation defines the rough machining of the external surfaces using an external roughing tool. The Rough option is chosen for the work type to perform the machining in a number of equidistant passes. Long external mode is used to create the passes above the geometry and along the rotational axis of the part. Non-descending motions strategy is used to avoid collisions.

- **Drilling (DRILL)**

  This Drilling operation defines the drilling of a hole in the center of the part to make machining the internal contour easier and more efficient. A Ø33 mm (1.375 in) flat drill is used.

- **Internal roughing (TR_contour2)**

  This Turning operation defines the rough machining of the internal surfaces using an internal roughing tool. The Rough option is chosen for the work type and Long internal mode is used to create passes under the geometry and along the rotational axis of the part.

- **Contour rest machining and finishing (TR_contour3; TR_contour3_1; TR_contour4; TR_contour8)**

  These Turning operations define the rest and finish machining of the external surfaces. In the first operation, an external roughing tool with different setup angle is used to remove unmachined material from the external contour. The Smooth option is chosen for the roughing type to provide a smoother finish. In the remaining operations, the Finish only option is chosen for the work type to perform only finishing cuts.

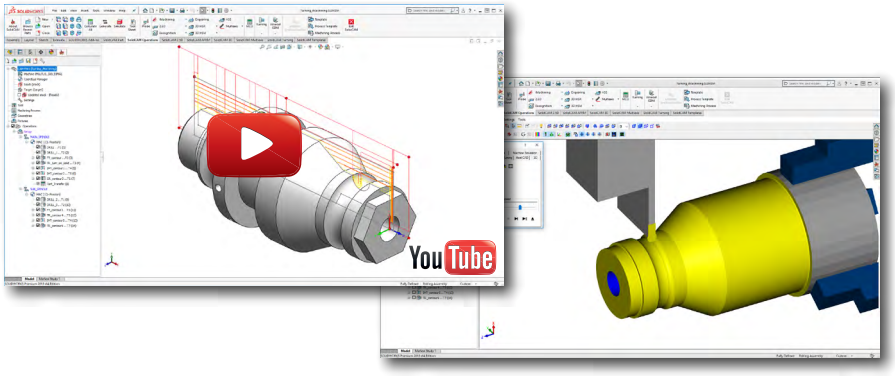
- **Internal rest machining (TR_contour6; TR_contour7)**

  These Turning operations define the rest machining of the internal surfaces. The first operation removes unmachined material from the internal contour. The Smooth option is chosen for the roughing type to provide a smoother finish. The second operation uses a different tool mounting orientation to perform roughing cuts towards the center of the part.

- **Internal finishing (TR_contour9; TR_contour7_1)**

  These Turning operations define the finish machining of the internal surfaces using the Finish only option. As in the previous operations, different tool mounting orientations are used to perform finishing cuts both away from and towards the center of the part.

Turning – 2 Spindles



Click to open the SolidCAM part: **Turning_iMachining.prz**

Click to download the recording.

The operations in this example illustrate the use of SolidCAM's Trochoidal Turning and several other Turning technologies with front and back spindles functionality to machine the part shown above.

- **Main spindle face drilling (DRILL; DRILL_1)**

  These Drilling operations define the drilling of the hole in the center of the front face.

- **Main spindle face turning (FT_contour)**

  This Face Turning operation defines the machining of the front face using an external roughing tool. The specified 0.2 mm (0.008 in) Z-Offset is removed in the same operation with a finishing cut.

- **Main spindle contour turning (TR_turn_on_solid)**

  This Turning operation defines the machining of the external surfaces on the front side using an external roughing tool. Long external mode is used and the Rough option is chosen for the work type. The Smooth option provides a smoother finish during the roughing, which is then followed by a finishing cut. Non-descending motions strategy is used to avoid collisions.

- **Main spindle round grooving (IMT_contour1)**

  This Trochoidal Turning operation defines the machining of the round groove using an external grooving tool with round insert. Long external mode is used and a Cutting angles range of 30° to 45° is specified. Zigzag is chosen for the direction to keep the tool in contact with the material and to make machining the groove more efficient.

- **Main spindle flat grooving (IMT_contour2; GR_contour2)**

  These operations define the machining of the flat groove.

  The Trochoidal Turning operation provides roughing of the groove using an external grooving tool with round insert. The same technological parameters are used as in the previous operation minus a finishing cut. When the tool reaches the bottom of the groove, iMachining movements are performed in the corners.

  The Grooving operation provides finishing of the groove using a typical grooving tool. The same geometry is used to remove the material remaining in the corners. The Finish only option is chosen for the work type to perform only finishing cuts on the rest material.

- **Sub-spindle face drilling (DRILL_2; DRILL_3)**

  These Drilling operations define the drilling of the hole in the center of the back face.

- **Sub-spindle face turning (FT_contour3)**

  This Face Turning operation defines the machining of the back face using an external roughing tool. The specified 0.2 mm (0.008 in) Z-Offset is removed in the same operation with a finishing cut.

- **Sub-spindle contour turning (TR_contour4)**

  This Turning operation defines the machining of the external surfaces on the back side using an external roughing tool. Long external mode is used and the Rough option is chosen for the work type. The Smooth option provides a smoother finish during the roughing, which is then followed by a finishing cut. Non-descending motions strategy is used to avoid collisions.

- **Sub-spindle grooving (IMT_contour5; TR_contour6)**

  These operations define the machining of the round and inclined grooves.

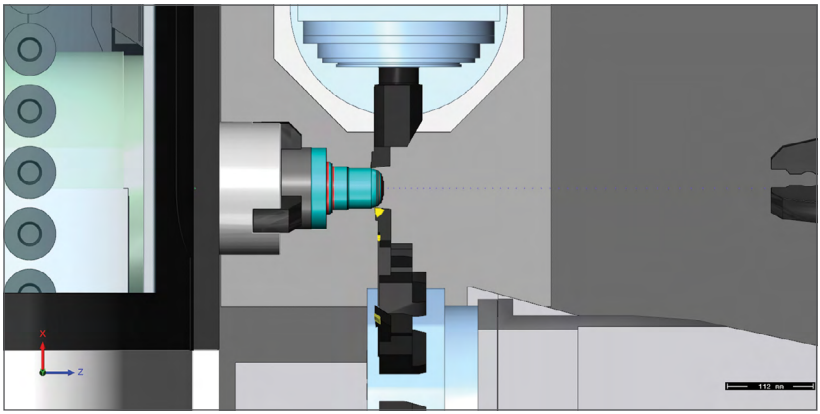
  The Trochoidal Turning operation provides the rough and finish machining of the round groove using an external grooving tool with round insert. Long external mode is used and a Cutting angles range of 30° to 45° is specified. Zigzag is chosen for the direction to keep the tool in contact with the material and to make machining the groove more efficient.

  The Turning operation provides the rough and finish machining of the inclined groove using an external roughing tool. Long external mode is used and the Rough option is chosen for the work type. The Smooth option provides a smoother finish during the roughing, which is then followed by a finishing cut.

Easy Programming for Complex Multiple Turret and Multiple Spindle CNC-Machines

The fastest growing and most demanding class of CNC-Machines on the market today are multi-task machines that combine several capabilities into one machine – multiple spindles, multiple turrets, material being machined in multiple stages, transferring from spindle to spindle without handling, stock being inserted at one end and finished parts coming out the other.

On a Mill-Turn CNC-Machine, C-, Y- and B-Axis milling and drilling occur within the same program as the turning, providing a fully integrated and associative programming solution.



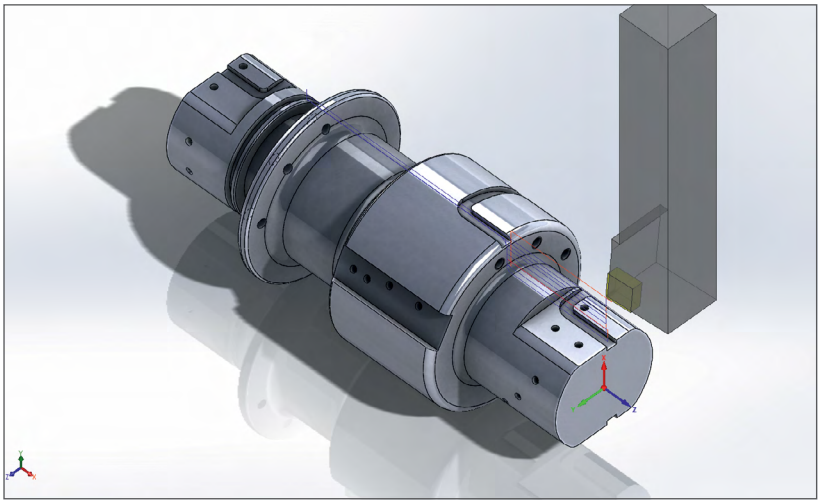
4/5-Axis Simultaneous Mill-Turn Machines have many uses and allow much more flexibility and capabilities not offered from other machine configurations. With this in mind, many of these have multiaxis, upper turrets, lower turrets, CYB and sub-spindles.

SolidCAM has the advanced technology to support the programming of all the latest multi-function CNC-Machines, providing powerful tools that are easy to learn and use, offering ultimate flexibility and configurability.

- Use of tail stocks, steady rest, sub-spindles, rotary and linear turrets along with C-Axis, Y-Axis and B-Axis, are regular features on today's Mill-Turn machine tools, and in this collision risky environment, the programming of these machines is made simple and safe with SolidCAM Turning and Milling in a single environment
- Support for multiple turret and multiple spindle programming with Multi-turret Synchronization and full Machine Simulation, seamlessly integrated into one extremely powerful package
- All SolidCAM Turning and Milling operations, including the unique, revolutionary iMachining, are available for the programming of Mill-Turn Machines, and all ancillary devices can also be defined and considered during simulation and gouge checking

Easy Programming for Complex Mill-Turn CNC-Machines

SolidCAM goes beyond just programming these complex machines with intelligent management of rest material between all operations, both Milling and Turning, for the most efficient tool paths and reduced cycle times to ensure the highest possible productivity ever imagined.

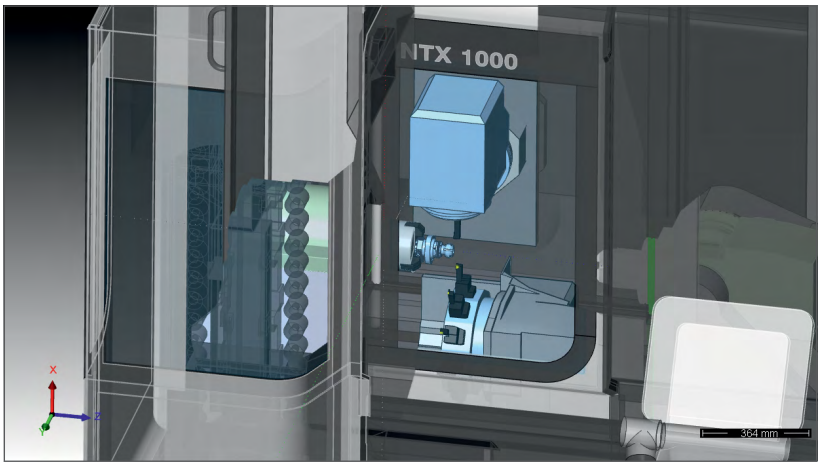


VMID (Virtual Machine ID)

The VMID defines the CNC-Machine components and machine kinematics, enabling users to set up and support the most complicated Mill-Turn Machines easily and effectively.

MCO (Machine Control Operation)

MCO (Machine Control Operation) enables the user to insert various control operations while manufacturing a part. These operations control the CNC-Machine and activate different options and devices such as the opening or closing of fixtures, activating coolants, rotating the part or moving the part from one table to another.

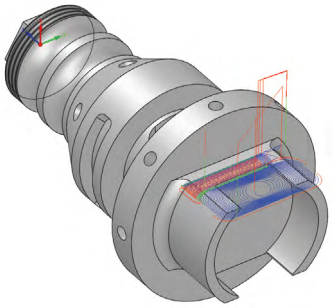


Transfer Between Spindles

Control the transfer of parts between the main and sub-spindle, using Machine Control Operations. Ready-made MCOs provide the best solution for this process.

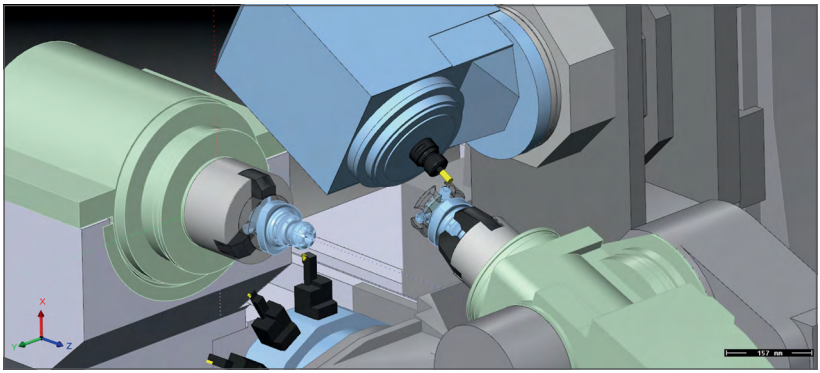
iMachining in Mill-Turn

Using iMachining 2D & 3D saves you both programming and cycle time. Additionally, iMachining has its very important, unmatched advantages of exerting smaller cutting forces, eliminating vibrations and excessive tool wear, even in situations of non-rigid workpiece holding.

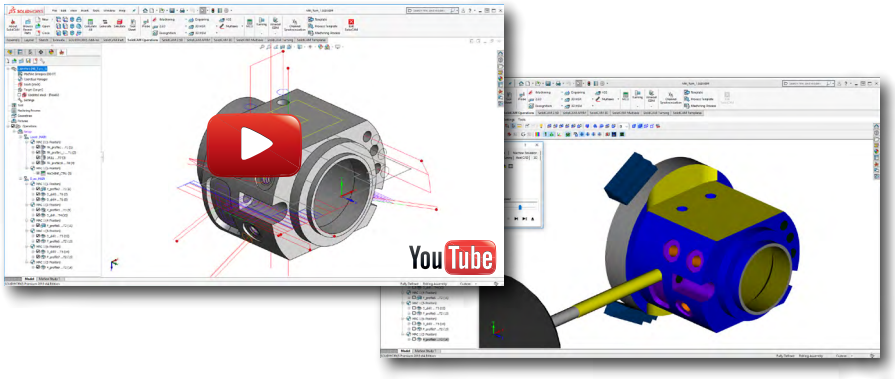


Mill-Turn Machine Simulation

Mill-Turn Machine Simulation in SolidCAM offers a full kinematic simulation package, supporting simulation of all Turning and Milling operations and of all CNC-Machine components and devices. The simulator offers full collision detection and avoidance between machine components, workpiece, fixtures and tool holders.



All the cycles and movements are supported along with the full graphics of the machine components and auxiliary devices such as tail stock and steady rest, providing safety as the part is fully tested before reaching the actual machine tool.



Click to open the SolidCAM part: **Mill_Turn_1.prz**

Click to download the recording.

The operations in this example illustrate the use of SolidCAM's Mill-Turn module to machine the part shown above. The machining is performed on a 5-Axis Mill-Turn CNC-Machine using several SolidCAM Milling and Turning technologies and Indexial Multiaxis Machining capabilities.

- **Contour and internal turning (TR_profile1; TR_profile1_1; DRILL; TR_profile10)**

  These Turning operations define the machining of the external and internal cylindrical surfaces using the appropriate tools, machining modes, work types, roughing types and finishing methods.

- **Facial milling (F_profile2; D_drill3; D_drill4)**

  These Milling operations define the machining of the screw slot and the drilling of the through hole and three blind holes using Machine Coordinate System #1, Position #1 (MAC 1 (1- Position)).

- **Side faces machining (P_profile3)**

  This Pocket operation defines the indexial machining of the side faces using MAC 1 (3- Position). The Contour strategy is combined with a negative Wall offset value to generate an overlapping tool path that completely machines the faces. The Transform option creates a circular pattern of operations around the revolution axis in order to machine both faces.

- **Side face drilling (D_drill)**

  This Drilling operation defines the indexial drilling of the two holes located on the side face using MAC 1 (3- Position).

- **Slot machining (F_profile5)**

  This Profile operation defines the indexial machining of the slot using MAC 1 (4- Position). The material is removed at a constant depth followed by an additional finish pass.

- **Radial holes machining**
  (D_drill1; P_profile6; D_drill2; P_profile7)

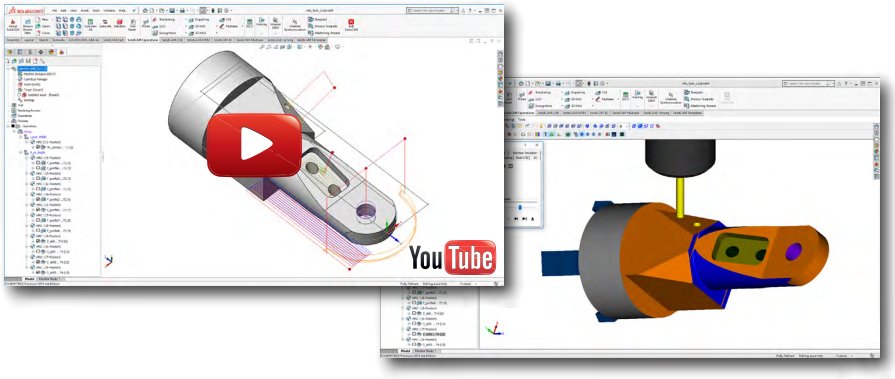
  These operations define the indexial machining of the three counterbore holes on the external cylindrical surface.

  The first Drilling and Profile operations use MAC 1 (5- Position) to perform the hole drilling and counterbore machining on one side of the screw slot.

  The following Drilling and Profile operations use MAC 1 (6- Position) to perform the holes drilling and counterbores machining on the other side of the screw slot.

- **Wrapped pocket machining (P_profile9)**

  This Pocket operation defines the simultaneous machining of the pocket on the external cylindrical surface using MAC 1 (2- Position). Wrapped is selected for the geometry type and the Contour strategy is used to perform the pocket machining.



Click to open the SolidCAM part: **Mill_Turn_2.prz**

Click to download the recording.

The operations in this example illustrate the use of SolidCAM's Mill-Turn module to machine the part shown above. The machining is performed on a 5-Axis Mill-Turn CNC-Machine using several SolidCAM Milling and Turning technologies and Indexial Multiaxis Machining capabilities.

- **Contour turning (TR_contour)**

  This Turning operation defines the machining of the cylindrical surface.

- **Cube machining (F_profile6)**

  This Profile operation defines the indexial machining of the cube faces at the base of the part using Machine Coordinate System #1, Position #4 (MAC 1 (4- Position)). The Transform option creates a circular pattern of operations around the revolution axis in order to machine all the faces.

- **Horizontal faces machining (F_profile1)**

  This Profile operation defines the indexial machining of the horizontal faces at the front of the part using MAC 1 (4- Position). The Transform option creates a circular pattern of operations around the revolution axis in order to machine both faces.

- **Inclined faces machining (F_profile3; F_profile4)**

  These Profile operations define the machining of the inclined faces that are adjacent to the previously machined horizontal faces using MAC 1 (5 and 6- Positions) and the B-Axis tilting capabilities of the tool.

- **Radial face machining (F_profile2)**

  This Profile operation defines the machining of the cylindrical face at the front of the part using MAC 1 (4- Position).

- **Inclined pocket machining (P_profile9)**

  This Pocket operation defines the machining of the pocket on the previously machined inclined face using MAC 1 (5- Position) and the B-Axis tilting capabilities of the tool.

- **Cube inclined faces machining (F_profile7; F_profile8)**

  These Profile operations define the machining of the inclined faces on the cube corners using MAC 1 (7 and 8- Positions) and the B-Axis tilting capabilities of the tool.

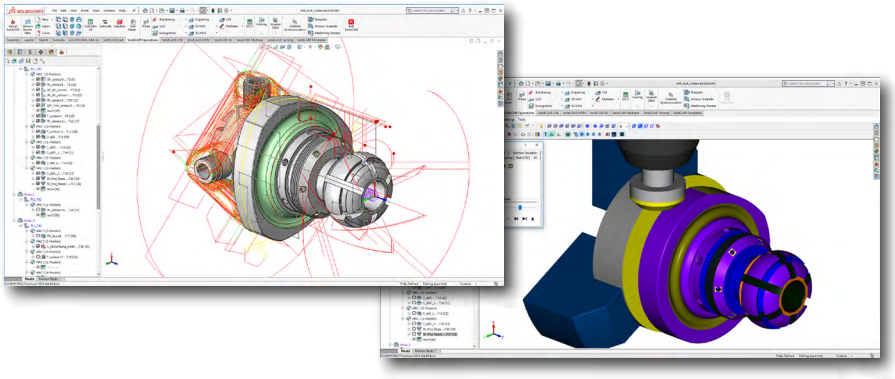
- **Through holes drilling (D_drill; D_drill1)**

  These Drilling operations define the drilling of the hole through the horizontal faces at the front of the part using MAC 1 (4- Position) and the drilling of the holes through the adjacent inclined faces using MAC 1 (6- Position) and the B-Axis tilting capabilities of the tool.

- **Cube inclined faces drilling (D_drill2; D_drill3)**

  These Drilling operations define the drilling of the holes on the cube inclined faces using MAC 1 (7 and 8- Positions) and the B-Axis tilting capabilities of the tool.

Advanced Mill-Turn



Click to open the SolidCAM part: **Mill_Turn_Advanced.prz**

The operations in this example illustrate the use of SolidCAM's Mill-Turn module to machine the part shown above. The machining is performed on a 5-Axis Mill-Turn CNC-Machine using several SolidCAM Milling and Turning technologies as well as Indexial Multiaxis and Simultaneous 5-Axis Machining capabilities. The following operations are included but not limited to:

- **Contour balanced roughing (TBR_contour5)**

  This Balanced Rough operation defines the rough machining of the external cylindrical surfaces on the front side using a Master submachine and Slave submachine to drive two tools at the same time. The Trailing option starts working one tool behind the other, each one turning a different diameter. Non-descending motions strategy is used to avoid collisions.

- **Front face turning (TR_contour7)**

  This Turning operation defines the rough machining of the front face using a tool in the lower turret. Face front mode is used to create a roughing pass along the front face of the part.

- **Contour rest machining and front face finishing (TR_contour33; CUT_contour32; TR_contour36)**

  These Turning operations define the rest machining of the small grooves on both sides of the thread feature and the finishing of the front face using the appropriate tools.

- **External threading (TH_contour2)**

  This Threading operation defines the machining of an external thread using an external threading tool. A user-defined Thread size and Pitch unit are specified with no thread finishing.

- **Adjusting turret position (mco2)**

  This Machine Control Operation (MCO) defines a change in the position of the lower turret. It enables the use of the upper turret with subsequent operations safely, without any interference.

- **Grooving (GR_contour8)**

  This Grooving operation defines the rough and finish machining of the large groove near the back side of the part using an external grooving tool.

- **Contour finishing (TR_contour9)**

  This Turning operation defines the finish machining of the small grooves on both sides of the thread feature using a typical grooving tool.

- **Angled grooving (GR_AC_contour; GR_AC_contour1)**

  These Angled Grooving operations define the rough and finish machining of the angled grooves around the bulb feature at the front of the part using typical grooving tools with additional angles defined.

- **Rough grooving and simultaneous finishing (TR_contour3; SIM_Turn_contour3)**

  These Turning operations define the rough and finish machining of the round groove on the front face of the middle part. Both operations use the same geometry.

- **Hole roughing and internal finishing (F_contour4; TR_contour11)**

  These Profile and Turning operations define the rough and finish machining of the hole in the center of the front face.

- **Slots machining (5X_Proj_faces)**

  This Sim. 5-Axis Milling operation uses the Projection (User-defined) strategy to define the machining of the six slots on the bulb feature at the front of the part.

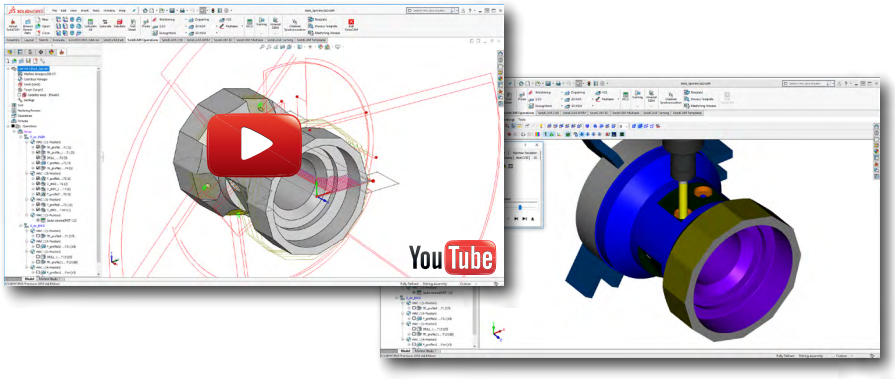
- **Middle part machining (5X_Proj_faces1)**

  This Sim. 5-Axis Milling operation uses the Projection (User-defined) strategy to define the machining of the three surfaces around the middle part.

- **Studs rough machining (G_i3DSurfacing_model)**

  This iMachining 3D operation defines the indexial roughing of the four studs at the back of the part using Machine Coordinate System #3, Position #2 (MAC 3 (2- Position)). The Transform option creates a circular pattern of operations around the revolution axis in order to rough all the studs.

Mill-Turn – 2 Spindles



Click to open the SolidCAM part: **Back_Spindle.prz**

Click to download the recording.

The operations in this example illustrate the use of SolidCAM's Mill-Turn module with front and back spindles functionality to machine the part shown above. The machining is performed on a 5-Axis Mill-Turn CNC-Machine using several SolidCAM Milling and Turning technologies and Indexial Multiaxis Machining capabilities.

- **Main spindle contour and internal turning (TR_profile; TR_profile_1; DRILL; TR_profile2)**

  These Turning operations define the machining of the external and internal cylindrical surfaces on the front side using the appropriate tools, machining modes, work types, roughing types and finishing methods.

- **Main spindle facial milling (F_profile1)**

  This Profile operation defines the machining of the facets around the external surface on the front side using Machine Coordinate System #1, Position #1 (MAC 1 (1- Position)). The material is removed at a constant depth followed by an additional finish pass.

- **Main spindle middle part pads machining (F_profile6)**

  This Profile operation defines the indexial machining of the pads around the cylindrical surface on the middle of the part using MAC 1 (5- Position). The Transform option creates a circular pattern of operations around the revolution axis in order to machine all the pads.

- **Main spindle middle part holes machining (D_drill2; D_drill2_1; F_profile7)**

  These operations define the indexial machining of the counterbore holes on the previously machined pads using MAC 1 (5- Position). The Transform option creates a circular pattern of operations around the revolution axis in order to machine all the holes and counterbores.

  The Drilling operations perform the center drilling and drilling of the holes.

  The Profile operation performs the counterbores machining.

- **Main spindle inclined pads machining/drilling (P_profile8; D_drill3)**

  These Profile and Drilling operations define the indexial machining/drilling of the pads and holes around the conical surface near the back side using MAC 1 (6- Position) and the B-Axis tilting capabilities of the tool. The Transform option creates a circular pattern of operations around the revolution axis in order to machine all the pads and holes.

- **Back spindle contour turning and facial milling (TR_profile9; F_profile10)**

  These Turning and Profile operations define the initial machining of the external cylindrical surface on the back side and the final machining of the facets around the external surface using MAC 1 (1 and 4- Positions).

- **Back spindle internal turning (DRILL_1; TR_profile11)**

  These Turning operations define the machining of the internal cylindrical surfaces on the back side using the appropriate tools, machining mode, work type, roughing type and finishing method.

- **Back spindle face machining (F_profile12; D_drill4; D_drill4_1)**

  These operations define the machining of the features on the back face using MAC 1 (4- Position).

  The Profile operation performs the ledge machining.

  The Drilling operations perform the center drilling and drilling of the holes on the ledge corners.

SolidCAM Essential Module for Tool Setup and On-the-Spot Measurement

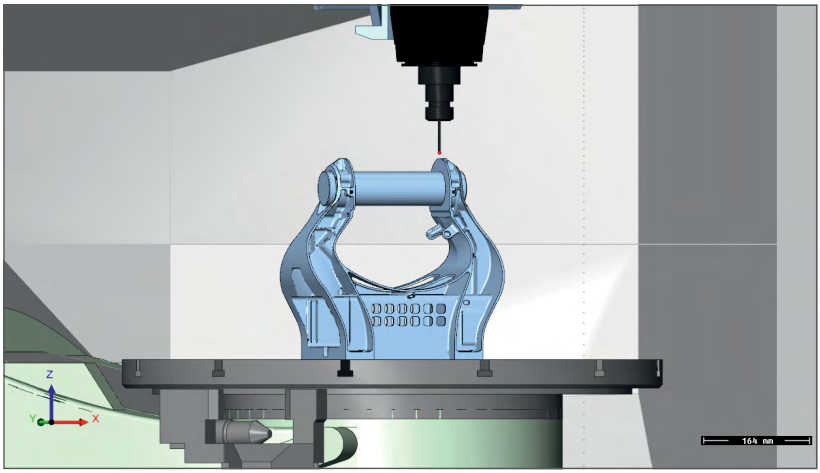
SolidCAM's integrated Solid Probe module provides Home Definition (Setup) and On-Machine Verification (Measurement) capabilities, using probes on the CNC-Machine.

Driving your probe with SolidCAM Solid Probe enables you to perform Setup as well as quick and easy Measurement for the best quality parts directly on your CNC-Machine, without breaking the Setup. Solid Probe also provides Tool Presetter support.

Full visualization of all the probe movements, provided by SolidCAM Machine Simulation, enables you to avoid any potential damage to the probe tool.

SolidCAM Solid Probe is a Must Module for Every Machinist Using Probes

SolidCAM's integrated Solid Probe module does it all.



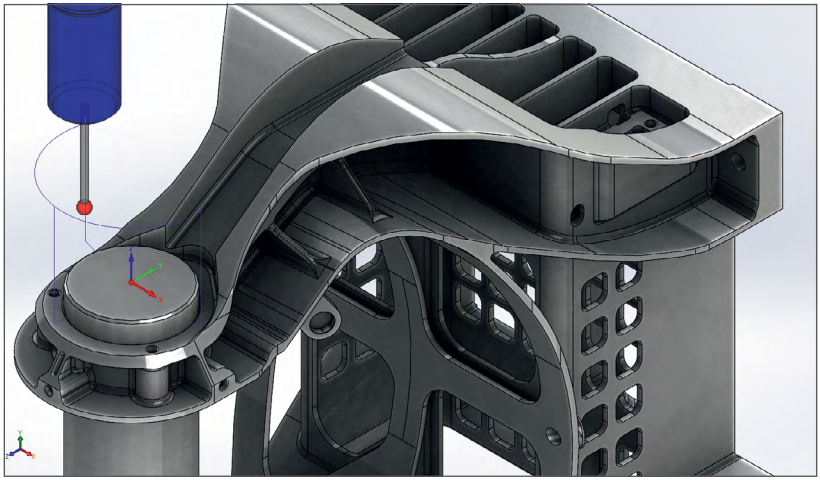
- Easy Home Definition
- Easy On-Machine Verification
- Tool Presetter support
- Easy geometry selection on solid model
- Supports wide range of Solid Probe Cycles
- Visualization of all probe tool movements
- Supports various controllers

Combined Probe and Machining Operations

Machining and Probe operations appear together in the SolidCAM Manager and can share the same geometries defined on the CAD model. When the solid model is changed, both Machining and Probe operations are automatically updated to use the new geometry. Many types of Solid Probe Cycles are provided to simplify measurement challenges and to smooth workflows alongside your Machining operations.

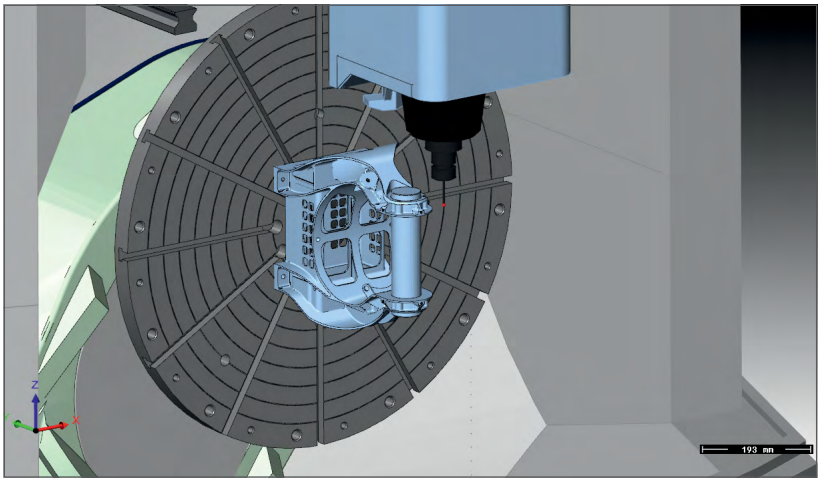
Home Definition

Solid Probe offers 16 different cycles to easily define your home positions, replacing manual Setup procedures that are often time consuming.



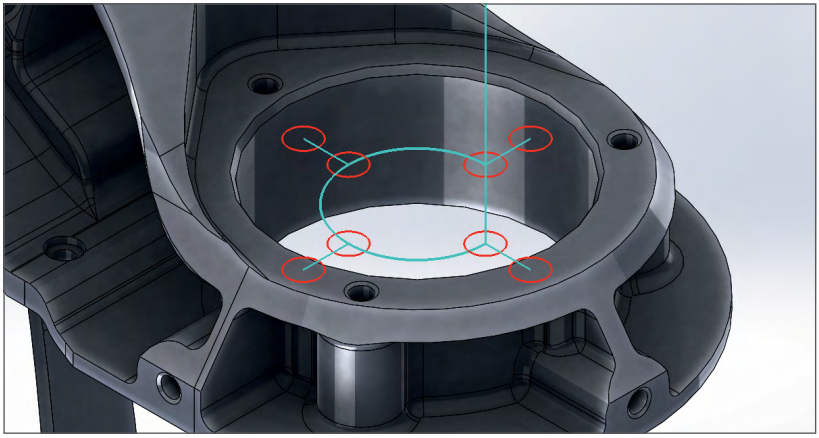
On-Machine Verification

Solid Probe Cycles are used for Measurement of machined surfaces without transferring the part to a Coordinate Measurement Machine (CMM) – the part can be inspected directly on the machine tool itself.



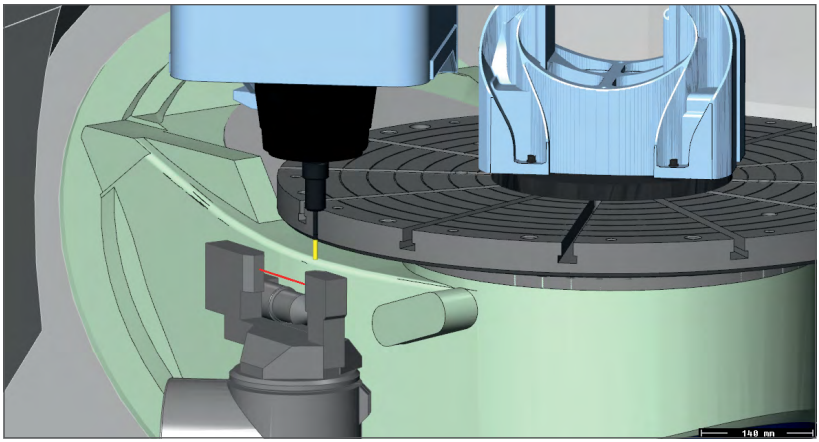
Preview of Cycle Movements

Solid Probe can use the same geometry as your 2.5D Milling operations with full control over tolerances and various sorting options and with a direct preview of the probe tool movements.

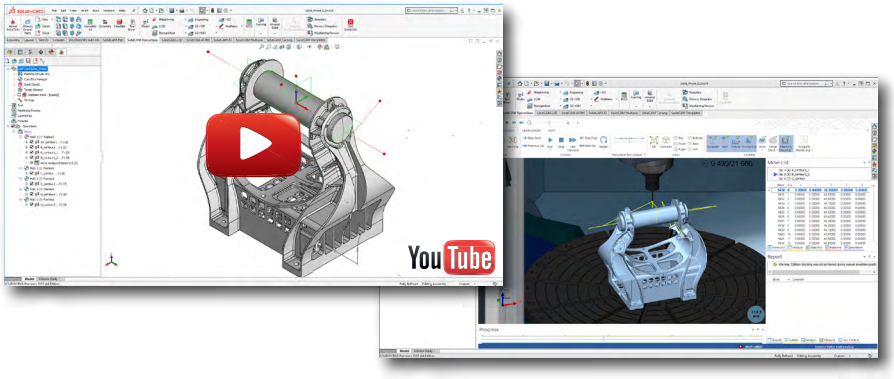


Tool Presetter Support

Solid Probe includes Tool Presetter support to check your milling and turning tools between Machining operations and tool change events. It also provides tool breakage detection to ensure continuous and safe machining.



Solid Probe



Click to open the SolidCAM part: **Solid_Probe.prz**

Click to download the recording.

The operations in this example illustrate the use of SolidCAM's Solid Probe Cycles to check the machining precision for different areas of the part shown above. One SolidCAM Coordinate System is defined with multiple positions.

- **Cylinder surface straightness checking (AY_contour1)**

  This Angle Y Cycle is defined to check if the cylinder surface is straight. Machine Coordinate System #1, Position #1 (MAC 1 (1- Position)) is used for the cycle.

- **Cylinder width checking (B_contour2)**

  This Boss Cycle is defined to check the width of the cylinder. MAC 1 (1- Position) is used for the cycle.

- **Cylinder length checking (B_contour1_1)**

  This Pocket Cycle is defined to check the length of the cylinder, measuring the distance between two boundary surfaces. MAC 1 (1- Position) is used for the cycle.

- **Cylinder height checking (B_contour1_2)**

  This Single point Z Cycle is used to check the height of the cylinder, measuring the position of the highest point. MAC 1 (1- Position) is used for the cycle.

- **Cylinder diameter checking (C_contour; C_contour3)**

  These Cylinder Cycles are used to check the diameters on both ends of the cylinder. MAC 1 (2 and 3- Positions) are used for the cycles.

- **Holes diameter checking (H_contour4; H_contour5)**

  These Hole Cycles are used to check the diameters of the holes on the tabs below the cylinder. MAC 1 (4 and 5- Positions) are used for the cycles.

System Requirements

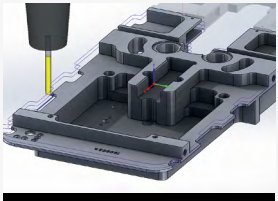
- Microsoft® Windows 7 (minimum 32-bit, 64-bit is recommended), Microsoft® Windows 8 64-bit, Microsoft® Windows 10 64-bit
- Intel® Core™2 Duo, or AMD dual-core processor with SSE2, 2 GHz or greater recommended, or Intel® Xeon® E3 or Core i7 or equivalent with SSE2, 3.0 GHz or greater
- 4 GB RAM (8 GB is recommended for large CAM-Parts)
- 15 GB free disk space for installation files
- 1,280 x 1,024 or higher screen resolution
- Mouse or other pointing device
- Microsoft® Direct3D 9® or compatible graphics card (Microsoft® Direct3D 11® or higher recommended)
- CD Drive
- Internet Explorer version 6 if you are viewing the SolidCAM Help Topics
- For viewing SolidCAM User Guides and Training Courses, Adobe Acrobat version 9 or higher is recommended

SolidCAM 2018

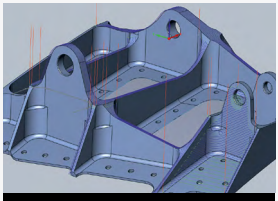
SolidCAM 2018 Modules Overview

Parts & Recordings

iMachining 2D

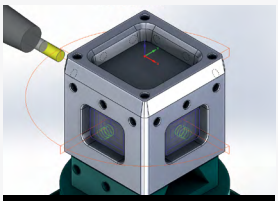


2.5D Milling



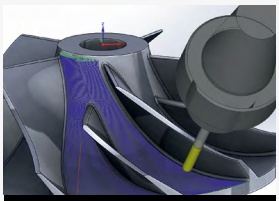
HSS (High-Speed Surface Machining)

iMachining 3D

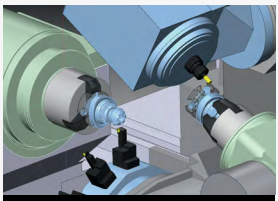


Indexial Multi-Sided Machining

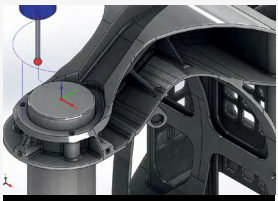
HSM (High-Speed Machining)



Sim. 5-Axis Machining



Turning & Advanced Mill-Turn



Solid Probe

youtube.com/SolidCAMProfessor
youtube.com/SolidCAMiMachining

facebook.com/SolidCAM
facebook.com/iMachining

iMachining Successes

solidcam.com/successes/imachining-successes

www.solidcam.com

SolidCAM
iMachining – The Revolution in CAM!